

UNCLASSIFIED

AD NUMBER

ADB017334

LIMITATION CHANGES

TO:

Approved for public release; distribution is unlimited.

FROM:

Distribution authorized to U.S. Gov't. agencies only; Test and Evaluation; DEC 1976. Other requests shall be referred to Air Force Flight Dynamics Lab., Wright-Patterson AFB, OH 45433.

AUTHORITY

AFWAL ltr 16 Sep 1982

THIS PAGE IS UNCLASSIFIED

AD

BO 17334

AUTHORITY:

AFWAL

11/16 SEP 82



(2)

PG

AD B017334

**ASOP-3: A PROGRAM FOR THE MINIMUM-WEIGHT DESIGN OF STRUCTURES
SUBJECTED TO STRENGTH AND DEFLECTION CONSTRAINTS**

**Grumman Aerospace Corporation
Bethpage, New York 11714**

December 1976

TECHNICAL REPORT AFFDL-TR-76-157

Final Report for Period 15 June 1975 – 15 December 1976

**Distribution limited to U.S. Government agencies only; test and evaluation; statement
applied December 15, 1976. Other requests for this document must be referred to
AF Flight Dynamics Laboratory, FBR, Wright-Patterson AFB, Ohio 45433.**

DDC
RECEIVED
MAR 31 1977
A

**AIR FORCE FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433**

AU NU.
DDC FILE COPY

NOTICE

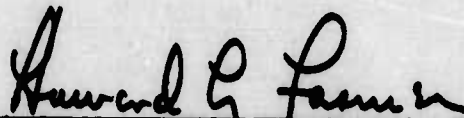
When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This technical report has been reviewed and is approved for publication.



V. B. VENKAYYA
Project Engineer

FOR THE COMMANDER



HOWARD L. FARMER, Col, USAF
Chief, Structural Mechanics Division
AF Flight Dynamics Laboratory

1. TITLE		2. DATE
3. AUTHORITY		4. STATUS
5. DISTRIBUTION		6. SPECIAL
7. REVISIONS		8. COMMENTS
9. APPROVAL		10. SIGNATURE
11. DISTRIBUTION		12. SPECIAL
13. REVISIONS		14. COMMENTS
15. APPROVAL		16. SIGNATURE
17. DISTRIBUTION		18. SPECIAL
19. REVISIONS		20. COMMENTS
21. APPROVAL		22. SIGNATURE
23. DISTRIBUTION		24. SPECIAL
25. REVISIONS		26. COMMENTS
27. APPROVAL		28. SIGNATURE
29. DISTRIBUTION		30. SPECIAL
31. REVISIONS		32. COMMENTS
33. APPROVAL		34. SIGNATURE
35. DISTRIBUTION		36. SPECIAL
37. REVISIONS		38. COMMENTS
39. APPROVAL		40. SIGNATURE
41. DISTRIBUTION		42. SPECIAL
43. REVISIONS		44. COMMENTS
45. APPROVAL		46. SIGNATURE
47. DISTRIBUTION		48. SPECIAL
49. REVISIONS		50. COMMENTS
51. APPROVAL		52. SIGNATURE
53. DISTRIBUTION		54. SPECIAL
55. REVISIONS		56. COMMENTS
57. APPROVAL		58. SIGNATURE
59. DISTRIBUTION		60. SPECIAL
61. REVISIONS		62. COMMENTS
63. APPROVAL		64. SIGNATURE
65. DISTRIBUTION		66. SPECIAL
67. REVISIONS		68. COMMENTS
69. APPROVAL		70. SIGNATURE
71. DISTRIBUTION		72. SPECIAL
73. REVISIONS		74. COMMENTS
75. APPROVAL		76. SIGNATURE
77. DISTRIBUTION		78. SPECIAL
79. REVISIONS		80. COMMENTS
81. APPROVAL		82. SIGNATURE
83. DISTRIBUTION		84. SPECIAL
85. REVISIONS		86. COMMENTS
87. APPROVAL		88. SIGNATURE
89. DISTRIBUTION		90. SPECIAL
91. REVISIONS		92. COMMENTS
93. APPROVAL		94. SIGNATURE
95. DISTRIBUTION		96. SPECIAL
97. REVISIONS		98. COMMENTS
99. APPROVAL		100. SIGNATURE
101. DISTRIBUTION		102. SPECIAL
103. REVISIONS		104. COMMENTS
105. APPROVAL		106. SIGNATURE
107. DISTRIBUTION		108. SPECIAL
109. REVISIONS		110. COMMENTS
111. APPROVAL		112. SIGNATURE
113. DISTRIBUTION		114. SPECIAL
115. REVISIONS		116. COMMENTS
117. APPROVAL		118. SIGNATURE
119. DISTRIBUTION		120. SPECIAL
121. REVISIONS		122. COMMENTS
123. APPROVAL		124. SIGNATURE
125. DISTRIBUTION		126. SPECIAL
127. REVISIONS		128. COMMENTS
129. APPROVAL		130. SIGNATURE
131. DISTRIBUTION		132. SPECIAL
133. REVISIONS		134. COMMENTS
135. APPROVAL		136. SIGNATURE
137. DISTRIBUTION		138. SPECIAL
139. REVISIONS		140. COMMENTS
141. APPROVAL		142. SIGNATURE
143. DISTRIBUTION		144. SPECIAL
145. REVISIONS		146. COMMENTS
147. APPROVAL		148. SIGNATURE
149. DISTRIBUTION		150. SPECIAL
151. REVISIONS		152. COMMENTS
153. APPROVAL		154. SIGNATURE
155. DISTRIBUTION		156. SPECIAL
157. REVISIONS		158. COMMENTS
159. APPROVAL		160. SIGNATURE
161. DISTRIBUTION		162. SPECIAL
163. REVISIONS		164. COMMENTS
165. APPROVAL		166. SIGNATURE
167. DISTRIBUTION		168. SPECIAL
169. REVISIONS		170. COMMENTS
171. APPROVAL		172. SIGNATURE
173. DISTRIBUTION		174. SPECIAL
175. REVISIONS		176. COMMENTS
177. APPROVAL		178. SIGNATURE
179. DISTRIBUTION		180. SPECIAL
181. REVISIONS		182. COMMENTS
183. APPROVAL		184. SIGNATURE
185. DISTRIBUTION		186. SPECIAL
187. REVISIONS		188. COMMENTS
189. APPROVAL		190. SIGNATURE
191. DISTRIBUTION		192. SPECIAL
193. REVISIONS		194. COMMENTS
195. APPROVAL		196. SIGNATURE
197. DISTRIBUTION		198. SPECIAL
199. REVISIONS		200. COMMENTS
201. APPROVAL		202. SIGNATURE
203. DISTRIBUTION		204. SPECIAL
205. REVISIONS		206. COMMENTS
207. APPROVAL		208. SIGNATURE
209. DISTRIBUTION		210. SPECIAL
211. REVISIONS		212. COMMENTS
213. APPROVAL		214. SIGNATURE
215. DISTRIBUTION		216. SPECIAL
217. REVISIONS		218. COMMENTS
219. APPROVAL		220. SIGNATURE
221. DISTRIBUTION		222. SPECIAL
223. REVISIONS		224. COMMENTS
225. APPROVAL		226. SIGNATURE
227. DISTRIBUTION		228. SPECIAL
229. REVISIONS		230. COMMENTS
231. APPROVAL		232. SIGNATURE
233. DISTRIBUTION		234. SPECIAL
235. REVISIONS		236. COMMENTS
237. APPROVAL		238. SIGNATURE
239. DISTRIBUTION		240. SPECIAL
241. REVISIONS		242. COMMENTS
243. APPROVAL		244. SIGNATURE
245. DISTRIBUTION		246. SPECIAL
247. REVISIONS		248. COMMENTS
249. APPROVAL		250. SIGNATURE
251. DISTRIBUTION		252. SPECIAL
253. REVISIONS		254. COMMENTS
255. APPROVAL		256. SIGNATURE
257. DISTRIBUTION		258. SPECIAL
259. REVISIONS		260. COMMENTS
261. APPROVAL		262. SIGNATURE
263. DISTRIBUTION		264. SPECIAL
265. REVISIONS		266. COMMENTS
267. APPROVAL		268. SIGNATURE
269. DISTRIBUTION		270. SPECIAL
271. REVISIONS		272. COMMENTS
273. APPROVAL		274. SIGNATURE
275. DISTRIBUTION		276. SPECIAL
277. REVISIONS		278. COMMENTS
279. APPROVAL		280. SIGNATURE
281. DISTRIBUTION		282. SPECIAL
283. REVISIONS		284. COMMENTS
285. APPROVAL		286. SIGNATURE
287. DISTRIBUTION		288. SPECIAL
289. REVISIONS		290. COMMENTS
291. APPROVAL		292. SIGNATURE
293. DISTRIBUTION		294. SPECIAL
295. REVISIONS		296. COMMENTS
297. APPROVAL		298. SIGNATURE
299. DISTRIBUTION		300. SPECIAL
301. REVISIONS		302. COMMENTS
303. APPROVAL		304. SIGNATURE
305. DISTRIBUTION		306. SPECIAL
307. REVISIONS		308. COMMENTS
309. APPROVAL		310. SIGNATURE
311. DISTRIBUTION		312. SPECIAL
313. REVISIONS		314. COMMENTS
315. APPROVAL		316. SIGNATURE
317. DISTRIBUTION		318. SPECIAL
319. REVISIONS		320. COMMENTS
321. APPROVAL		322. SIGNATURE
323. DISTRIBUTION		324. SPECIAL
325. REVISIONS		326. COMMENTS
327. APPROVAL		328. SIGNATURE
329. DISTRIBUTION		330. SPECIAL
331. REVISIONS		332. COMMENTS
333. APPROVAL		334. SIGNATURE
335. DISTRIBUTION		336. SPECIAL
337. REVISIONS		338. COMMENTS
339. APPROVAL		340. SIGNATURE
341. DISTRIBUTION		342. SPECIAL
343. REVISIONS		344. COMMENTS
345. APPROVAL		346. SIGNATURE
347. DISTRIBUTION		348. SPECIAL
349. REVISIONS		350. COMMENTS
351. APPROVAL		352. SIGNATURE
353. DISTRIBUTION		354. SPECIAL
355. REVISIONS		356. COMMENTS
357. APPROVAL		358. SIGNATURE
359. DISTRIBUTION		360. SPECIAL
361. REVISIONS		362. COMMENTS
363. APPROVAL		364. SIGNATURE
365. DISTRIBUTION		366. SPECIAL
367. REVISIONS		368. COMMENTS
369. APPROVAL		370. SIGNATURE
371. DISTRIBUTION		372. SPECIAL
373. REVISIONS		374. COMMENTS
375. APPROVAL		376. SIGNATURE
377. DISTRIBUTION		378. SPECIAL
379. REVISIONS		380. COMMENTS
381. APPROVAL		382. SIGNATURE
383. DISTRIBUTION		384. SPECIAL
385. REVISIONS		386. COMMENTS
387. APPROVAL		388. SIGNATURE
389. DISTRIBUTION		390. SPECIAL
391. REVISIONS		392. COMMENTS
393. APPROVAL		394. SIGNATURE
395. DISTRIBUTION		396. SPECIAL
397. REVISIONS		398. COMMENTS
399. APPROVAL		400. SIGNATURE
401. DISTRIBUTION		402. SPECIAL
403. REVISIONS		404. COMMENTS
405. APPROVAL		406. SIGNATURE
407. DISTRIBUTION		408. SPECIAL
409. REVISIONS		410. COMMENTS
411. APPROVAL		412. SIGNATURE
413. DISTRIBUTION		414. SPECIAL
415. REVISIONS		416. COMMENTS
417. APPROVAL		418. SIGNATURE
419. DISTRIBUTION		420. SPECIAL
421. REVISIONS		422. COMMENTS
423. APPROVAL		424. SIGNATURE
425. DISTRIBUTION		426. SPECIAL
427. REVISIONS		428. COMMENTS
429. APPROVAL		430. SIGNATURE
431. DISTRIBUTION		432. SPECIAL
433. REVISIONS		434. COMMENTS
435. APPROVAL		436. SIGNATURE
437. DISTRIBUTION		438. SPECIAL
439. REVISIONS		440. COMMENTS
441. APPROVAL		442. SIGNATURE
443. DISTRIBUTION		444. SPECIAL
445. REVISIONS		446. COMMENTS
447. APPROVAL		448. SIGNATURE
449. DISTRIBUTION		450. SPECIAL
451. REVISIONS		452. COMMENTS
453. APPROVAL		454. SIGNATURE
455. DISTRIBUTION		456. SPECIAL
457. REVISIONS		458. COMMENTS
459. APPROVAL		460. SIGNATURE
461. DISTRIBUTION		462. SPECIAL
463. REVISIONS		464. COMMENTS
465. APPROVAL		466. SIGNATURE
467. DISTRIBUTION		468. SPECIAL
469. REVISIONS		470. COMMENTS
471. APPROVAL		472. SIGNATURE
473. DISTRIBUTION		474. SPECIAL
475. REVISIONS		476. COMMENTS
477. APPROVAL		478. SIGNATURE
479. DISTRIBUTION		480. SPECIAL
481. REVISIONS		482. COMMENTS
483. APPROVAL		484. SIGNATURE
485. DISTRIBUTION		486. SPECIAL
487. REVISIONS		488. COMMENTS
489. APPROVAL		490. SIGNATURE
491. DISTRIBUTION		492. SPECIAL
493. REVISIONS		494. COMMENTS
495. APPROVAL		496. SIGNATURE
497. DISTRIBUTION		498. SPECIAL
499. REVISIONS		500. COMMENTS
501. APPROVAL		502. SIGNATURE
503. DISTRIBUTION		504. SPECIAL
505. REVISIONS		506. COMMENTS
507. APPROVAL		508. SIGNATURE
509. DISTRIBUTION		510. SPECIAL
511. REVISIONS		512. COMMENTS
513. APPROVAL		514. SIGNATURE
515. DISTRIBUTION		516. SPECIAL
517. REVISIONS		518. COMMENTS
519. APPROVAL		520. SIGNATURE
521. DISTRIBUTION		522. SPECIAL
523. REVISIONS		524. COMMENTS
525. APPROVAL		526. SIGNATURE
527. DISTRIBUTION		528. SPECIAL
529. REVISIONS		530. COMMENTS
531. APPROVAL		532. SIGNATURE
533. DISTRIBUTION		534. SPECIAL
535. REVISIONS		536. COMMENTS
537. APPROVAL		538. SIGNATURE
539. DISTRIBUTION		540. SPECIAL
541. REVISIONS		542. COMMENTS
543. APPROVAL		544. SIGNATURE
545. DISTRIBUTION		546. SPECIAL
547. REVISIONS		548. COMMENTS
549. APPROVAL		550. SIGNATURE
551. DISTRIBUTION		552. SPECIAL
553. REVISIONS		554. COMMENTS
555. APPROVAL		556. SIGNATURE
557. DISTRIBUTION		558. SPECIAL
559. REVISIONS		560. COMMENTS
561. APPROVAL		562. SIGNATURE
563. DISTRIBUTION		564. SPECIAL
565. REVISIONS		566. COMMENTS
567. APPROVAL		568. SIGNATURE
569. DISTRIBUTION		570. SPECIAL
571. REVISIONS		572. COMMENTS
573. APPROVAL		574. SIGNATURE
575. DISTRIBUTION		576. SPECIAL
577. REVISIONS		578. COMMENTS
579. APPROVAL		580. SIGNATURE
581. DISTRIBUTION		582. SPECIAL
583. REVISIONS		584. COMMENTS
585. APPROVAL		586. SIGNATURE
587. DISTRIBUTION		588. SPECIAL
589. REVISIONS		590. COMMENTS
591. APPROVAL		592. SIGNATURE
593. DISTRIBUTION		594. SPECIAL
595. REVISIONS		596. COMMENTS
597. APPROVAL		598. SIGNATURE
599. DISTRIBUTION		600. SPECIAL
601. REVISIONS		602. COMMENTS
603. APPROVAL		604. SIGNATURE
605. DISTRIBUTION		606. SPECIAL
607. REVISIONS		608. COMMENTS
609. APPROVAL		610. SIGNATURE
611. DISTRIBUTION		612. SPECIAL
613. REVISIONS		614. COMMENTS
615. APPROVAL		616. SIGNATURE
617. DISTRIBUTION		618. SPECIAL
619. REVISIONS		620. COMMENTS
621. APPROVAL		622. SIGNATURE
623. DISTRIBUTION		624. SPECIAL
625. REVISIONS		626. COMMENTS
627. APPROVAL		628. SIGNATURE
629. DISTRIBUTION		630. SPECIAL
631. REVISIONS		632. COMMENTS
633. APPROVAL		634. SIGNATURE
635. DISTRIBUTION		636. SPECIAL
637. REVISIONS		638. COMMENTS
639. APPROVAL		640. SIGNATURE
641. DISTRIBUTION		642. SPECIAL
643. REVISIONS		644. COMMENTS
645. APPROVAL		646. SIGNATURE
647. DISTRIBUTION		648. SPECIAL
649. REVISIONS		650. COMMENTS
651. APPROVAL		652. SIGNATURE
653. DISTRIBUTION		654. SPECIAL
655. REVISIONS		656. COMMENTS
657. APPROVAL		658. SIGNATURE
659. DISTRIBUTION		660. SPECIAL
661. REVISIONS		662. COMMENTS
663. APPROVAL		664. SIGNATURE
665. DISTRIBUTION		666. SPECIAL
667. REVISIONS		668. COMMENTS
669. APPROVAL		670. SIGNATURE
671. DISTRIBUTION		672. SPECIAL
673. REVISIONS		674. COMMENTS
675. APPROVAL		676. SIGNATURE
677. DISTRIBUTION		678. SPECIAL
679. REVISIONS		680. COMMENTS
681. APPROVAL		682. SIGNATURE
683. DISTRIBUTION		684. SPECIAL
685. REVISIONS		686. COMMENTS
687. APPROVAL		688. SIGNATURE
689. DISTRIBUTION		690. SPECIAL
691. REVISIONS		692. COMMENTS
693. APPROVAL		694. SIGNATURE
695. DISTRIBUTION		696. SPECIAL
697. REVISIONS		698. COMMENTS
699. APPROVAL		700. SIGNATURE
701. DISTRIBUTION		702. SPECIAL
703. REVISIONS		704. COMMENTS
705. APPROVAL		706. SIGNATURE
707. DISTRIBUTION		708. SPECIAL
709. REVISIONS		710. COMMENTS
711. APPROVAL		712. SIGNATURE
713. DISTRIBUTION		714. SPECIAL
715. REVISIONS		716. COMMENTS
717. APPROVAL		718. SIGNATURE
719. DISTRIBUTION		720. SPECIAL
721. REVISIONS		722. COMMENTS
723. APPROVAL		724. SIGNATURE
725. DISTRIBUTION		726. SPECIAL
727. REVISIONS		728. COMMENTS
729. APPROVAL		730. SIGNATURE
731. DISTRIBUTION		732. SPECIAL
733. REVISIONS		734. COMMENTS
735. APPROVAL		736. SIGNATURE
737. DISTRIBUTION		738. SPECIAL
739. REVISIONS		740. COMMENTS
741. APPROVAL		742. SIGNATURE
743. DISTRIBUTION		744. SPECIAL
745. REVISIONS		746. COMMENTS
747. APPROVAL		748. SIGNATURE
749. DISTRIBUTION		750. SPECIAL
751. REVISIONS		752. COMMENTS
753. APPROVAL		754. SIGNATURE
755. DISTRIBUTION		756. SPECIAL
757. REVISIONS		758. COMMENTS
759. APPROVAL		760. SIGNATURE
761. DISTRIBUTION		762. SPECIAL
763. REVISIONS		764. COMMENTS
765. APPROVAL		766. SIGNATURE
767. DISTRIBUTION		768. SPECIAL
769. REVISIONS		770. COMMENTS
771. APPROVAL		772. SIGNATURE
773. DISTRIBUTION		774. SPECIAL
775. REVISIONS		776. COMMENTS
777. APPROVAL		778. SIGNATURE
779. DISTRIBUTION		780. SPECIAL
781. REVISIONS		782. COMMENTS
783. APPROVAL		784. SIGNATURE
785. DISTRIBUTION		786. SPECIAL
787. REVISIONS		788. COMMENTS
789. APPROVAL		790. SIGNATURE
791. DISTRIBUTION		792. SPECIAL
793. REVISIONS		794. COMMENTS
795. APPROVAL		796. SIGNATURE
797. DISTRIBUTION		798. SPECIAL
799. REVISIONS		800. COMMENTS
801. APPROVAL		802. SIGNATURE
803. DISTRIBUTION		804. SPECIAL
805. REVISIONS		806. COMMENTS
807. APPROVAL		808. SIGNATURE
809. DISTRIBUTION		810. SPECIAL
811. REVISIONS		812. COMMENTS
813. APPROVAL		814. SIGNATURE
815. DISTRIBUTION		816. SPECIAL
817. REVISIONS		818. COMMENTS
819. APPROVAL		820. SIGNATURE
821. DISTRIBUTION		822. SPECIAL
823. REVISIONS		824. COMMENTS
825. APPROVAL		826. SIGNATURE
827. DISTRIBUTION		828. SPECIAL
829. REVISIONS		830. COMMENTS
831. APPROVAL		832. SIGNATURE
833. DISTRIBUTION		834. SPECIAL
835. REVISIONS		8

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM	
1. REPORT NUMBER	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER	
4. TITLE (and Subtitle)	5. TYPE OF REPORT & PERIOD COVERED	6. PERFORMING ORG. REPORT NUMBER	
7. AUTHOR(s)	8. CONTRACT OR GRANT NUMBER(s)	9. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS	
10. CONTROLLING OFFICE NAME AND ADDRESS	11. REPORT DATE	12. NUMBER OF PAGES	
13. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)	14. SECURITY CLASS. (of this report)	15a. DECLASSIFICATION/DOWNGRADING SCHEDULE	
16. DISTRIBUTION STATEMENT (of this Report)			
17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)			
18. SUPPLEMENTARY NOTES			
19. KEY WORDS (Continue on reverse side if necessary and identify by block number)			
20. ABSTRACT (Continue on reverse side if necessary and identify by block number)			

DD FORM 1473

1 JAN 73

EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

388 847

4B

CONF


UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)

The following improvements have been made:

- Composite laminates with as many as six fiber directions, which may be arbitrary, can be accommodated in both stress and deflection-constraint resizing.
- In stress-constraint resizing, a laminate element is treated as a unit, and criteria consistent with current design practice for composites are used.
- A deflection constraint generalized as a linear combination of translational displacements at several structural nodes can be applied. This permits the treatment of constraints on lifting-surface twist and camber.
- Multiple deflection constraints can be treated in a limited, but practically important, class of problems by multiple submissions of single-constraint cases.
- The program output has been extensively revised to make it more readable.

Results obtained by application of the program to models of two sample structures, a swept wing and a bomber fin, are presented.



FOREWORD

This report was prepared by the Structural Mechanics Section of the Grumman Aerospace Corporation, Bethpage, New York. It covers the further development of automated methods for structural optimization, including computer program development, and provides program user information. The work was performed under USAF Contract No. F33615-75-C-3146, Extension of Automated Structural Optimization Program (ASOP), which was initiated under Project No. 1467, Structural Analysis Methods, Task No. 146702, Work Unit No. 146702-37, Structural Optimization Methods for Aerospace Vehicles. The effort was administered by the Design and Analysis Branch of the Structural Mechanics Division, Air Force Flight Dynamics Laboratory, Air Force Systems Command, Wright-Patterson Air Force Base, Ohio. Dr. Vippera B. Venkayya (AFFDL/FBR) was the project monitor. The report covers work conducted between 15 June 1975 and 15 December 1976.

The project engineer and principal investigator for this work was Dr. Gabriel Isakson, Structural Methods Engineer, Structural Mechanics Section.

The authors gratefully acknowledge the continued interest and advice of Dr. Warner Lansing, Head of Structural Mechanics. They are also indebted to Mr. Edwin Lerner for valuable guidance and to Prof. Walter J. Dwyer of the University of Texas, who was the project engineer in the development of earlier versions of the ASOP program, for his help in gaining an understanding of the program. Other individuals who provided valuable aid and counsel are Dr. Joel Markowitz and Messrs. Dino George and Philip Stylianos.

Table of Contents

Section		Page
1	INTRODUCTION AND SUMMARY.	1
2	DESCRIPTION OF PROGRAM	5
2.1	Introduction	5
2.2	Structural Analysis and Modeling	6
2.3	The Stress-Constraint Mode	12
2.3.1	Basic Procedure and Element Stress Determination . .	12
2.3.2	Resizing Algorithm for Noncomposite Elements	14
2.3.3	Resizing Algorithm for Composite Elements	17
2.3.4	Resizing of Compression Panels	22
2.4	The Deflection-Constraint Mode	23
2.4.1	Resizing Algorithm	23
2.4.2	Resizing Algorithm for Interacting Deflection, Stress, and Minimum/Maximum Gage Constraints . . .	27
2.4.3	Treatment of Multiple Deflection Constraints as a Succession of Single-Constraint Problems	34
2.5	Flexible Supports	35
3	APPLICATIONS TO REPRESENTATIVE PROBLEMS	36
3.1	Introduction	36
3.2	Intermediate-Complexity Wing	36
3.3	Bomber Fin	41
4	USER INFORMATION	46
4.1	General Description and Limitations	46
4.2	Card Input and Illustrations of Data Preparation	47
4.2.1	General Construction of Input Deck	47
4.2.2	Initialization Card, Allowable Stress Modification Factors, Noncandidate Members for Deflection- Constraint Resizing	50
4.2.3	Nodal Geometry and Boundary Conditions	54
4.2.4	Applied Loads	59

Table of Contents (Continued)

Section		Page
4.2.5	Material Properties	61
4.2.6	Member Data	67
4.2.7	Deflection-Constraint Data	74
4.2.8	Stability Tables	78
4.2.9	Flexible Supports.	78
4.3	Description of Output	78
4.3.1	General Information	78
4.3.2	Input Data	79
4.3.3	Cyclic Data in the Stress-Constraint Mode	82
4.3.4	Cyclic Data in the Deflection-Constraint Mode	83
4.3.5	Final Data	86
4.4	Cautionary Notes	91
Appendix		
A	DESCRIPTION OF FINITE ELEMENTS	A-1
A.1	Introduction	A-1
A.2	Element No. 1 (Bar)	A-1
A.3	Element No. 2 (Beam).	A-2
A.4	Element No. 4 (Plane Stress Triangle)	A-3
A.5	Element No. 5 (Plane Stress Quadrilateral)	A-5
A.6	Element No. 8 (Warped Quadrilateral)	A-6
A.7	Element No. 6 (Warped Shear Panel)	A-9
A.8	Element No. 11 (Hinged Beam).	A-13
B	INPUT DATA FOR FINITE ELEMENTS	B-1
C	COMPUTATION OF ANGLE BETWEEN LOCAL ELEMENT AXES AND PROPERTY AXES	C-1
D	SUPPLEMENTARY MATHEMATICAL DETAILS PERTAINING TO STRUCTURAL ANALYSIS	D-1
D.1	Modified Cholesky Algorithm	D-1
D.2	Coordinate Transformations	D-2
D.3	Stiffness Properties of Composite Elements	D-4

Table of Contents (Continued)

Appendix		Page
E	DETERMINATION OF AVERAGE STRESS RESULTANTS IN TRIANGULAR AND QUADRILATERAL ELEMENTS	E-1
E.1	Determination of Average Stress Resultants in Triangular Elements	E-1
E.2	Determination of Average Stress Resultants in Quadrilateral Elements	E-3
F	MATHEMATICAL RELATIONS USED IN STRESS-CONSTRAINT RESIZING OF COMPOSITES	F-1
F.1	Determination of Stress Ratio for Composite-Element Layers	F-1
F.2	Application of Microbuckling Failure Criterion	F-3
G	DERIVATION OF MATHEMATICAL RELATIONS AND CRITERIA USED IN THE DEFLECTION-CONSTRAINT ALGORITHM	G-1
G.1	Uniform Derivatives as an Optimality Criterion in Deflection-Constraint Resizing	G-1
G.2	The Recurrence Relation in Deflection-Constraint Resizing . . .	G-3
G.3	Determination of Deflection Derivatives	G-5
G.4	Treatment of a Violated Inequality Constraint as an Equality Constraint	G-7
H	LIST OF PROGRAM SUBROUTINES	H-1
I	OVERLAY CHARTS	I-1
	REFERENCES	R-1

List of Illustrations

Figure		Page
1	Member Axis Systems.	10
2	Failure Criterion for Orthotropic Materials in Biaxial Stress	16
3	Cyclic Process for Stress-Constraint Resizing of Composite Elements	20
4	Iteration Cycles in Deflection-Constraint Mode	28
5	Algorithm for Deflection-Constraint Resizing	30
6	Aerodynamic Planform and Primary Structural Arrangement of Intermediate-Complexity Wing	37
7	Intermediate-Complexity Wing - Streamwise Twist Distribution at Various Stages of Resizing	39
8	Intermediate-Complexity Wing - Resizing History from Fully-Stressed Design to Deflection-Constrained Design for Inboard Station	40
9	Cover-Skin Layups for Intermediate-Complexity Wing	42
10	Substructure Gages for Intermediate-Complexity Wing	43
11	Bomber Fin Model and Cover-Skin Layups	44
12	Construction of Input Deck.	48
13	Initialization Card	51
14	Format for Allowable Stress Modification Factors and Noncandidate Members for Deflection-Constraint Resizing	55
15	Geometry and Boundary-Condition Format	56
16	Condensed Boundary Condition Format	58
17	Format for Applied Loads Data.	60
18	Format for Material Properties	64
19	Format for Coordinates of Points Defining Reference Directions for Property Axes	68
20	Standard Form for Member Data Card Preparation	69
21	Format for Material Properties and Minimum/Maximum Gages on Member Data Cards	72
22	Format for Deflection-Constraint Data	75

List of Illustrations (Continued)

Figure		Page
A-1	Bar Element	A-1
A-2	Beam Element	A-3
A-3	Triangular Membrane Element	A-4
A-4	Force Output for Triangular Membrane Element	A-4
A-5	Quadrilateral Membrane Element	A-5
A-6	Force Output for Quadrilateral Membrane Element	A-6
A-7	Warped Quadrilateral Element	A-7
A-8	Force Output for Warped Quadrilateral Element	A-8
A-9	Projected Quadrilateral	A-9
A-10	Equilibrium of Warped Shear Panel	A-10
A-11	Additional Geometry for Warped Shear Panel	A-11
C-1	Definition of Angle β	C-1
D-1	Element Axis Systems	D-3
D-2	Axis Systems for Layers of a Composite Element	D-6
E-1	Triangular-Element Geometry and Corner Forces	E-1
E-2	Quadrilateral-Element Geometry and Corner Forces	E-3
E-3	Forces on Quadrilateral Element Cross Sections	E-4
E-4	Transformation from Oblique to Rectangular Coordinates	E-8
G-1	Simple Structure Subjected to Angular Displacement Constraint	G-2
G-2	Design Space for Structure of Figure G-1	G-3
G-3	Satisfaction of an Inequality Deflection Constraint	G-8
I-1	Overlay Chart for ASOP-3 in IBM Systems	I-2
I-2	Overlay Chart for ASOP-3 in CDC Systems	I-3

List of Tables

Table		Page
1	List of Elements in ASOP-3	7
2	Summary of Results for Intermediate-Complexity Wing	41
3	Properties of Standard Materials	62

Section 1

INTRODUCTION AND SUMMARY

Full realization of the potential of advanced composite materials, for efficient aerospace structural design from a strength standpoint, and for the achievement of beneficial passive deformation in lifting surfaces, requires the availability of powerful and convenient automated design tools. Recent efforts in the development of the series of computer programs, under the label ASOP (Automated Structural Optimization Program), have been directed toward that end.

All of the ASOP programs, which operate on a finite-element model of the structure, apply the approach known as fully-stressed design for the satisfaction of strength requirements. In this approach, the structural members are sized so that every member is stressed to a permissible limit, in at least one loading condition, or is at a specified minimum gage. In the case of statically determinate structures, an optimum (minimum-weight) design can be arrived at directly. However, in the case more common in aerospace structures, that of highly redundant structures, an iterative procedure is necessary, with successive resizing of the members of the structure until a fully-stressed design is achieved. It has been demonstrated that such a design is not necessarily optimum from a weight standpoint (Reference 1), but it is believed to be close to optimum in most practical cases.

The ASOP-1 program (Reference 2) is limited to noncomposite structures, but includes a capability for treating translational displacement constraints. The ASOP-2 program is a further development (Reference 3) incorporating modifications to improve computational efficiency and the form of the results for useful application, as well as introducing a more generalized beam element and a capability for taking account of buckling in the strength resizing of noncomposite members. In addition, it can treat elastic constraints at support points and provides a means for estimating material reinforcement for bolted joints. Most importantly, it includes a capability for the strength resizing of laminated filamentary composites of balanced $0^\circ/90^\circ/\pm 45^\circ$ layup.

Both ASOP-1 and ASOP-2 use a method for determining stresses, known as the "nodal stress method," that tends to make the iterative process in fully-stressed design converge to a design in which structural properties vary more smoothly than when element average stresses are used (Reference 4). While this method works reasonably well for metal structures, particularly where the finite-element model has the form of a nearly-rectangular grid, it has been found that it can produce a distorted picture of the stresses and corresponding strains around a node when the grid is less regular, especially where triangular elements are used, and where adjacent elements are of dissimilar materials or substantially different thicknesses. This deficiency is particularly evident when composite materials are used.

ASOP-3, which is described in the present report, represents a further development in the ASOP series of programs. Because of the deficiency in the nodal stress method mentioned above, the program now utilizes element average stresses for resizing purposes. In the case of bars and triangular membrane elements, which are uniform-strain elements, the average stress is simply the uniform stress associated with the strain. In the case of quadrilateral membrane elements and shear panels, a heuristic procedure based on equilibrium is used to determine average stress. In beam elements, only that portion of the stress associated with axial load is an average stress; the bending stress is permitted to vary along the element and its maximum value is readily determined.

Where ASOP-2 is limited to balanced $0^\circ/90^\circ/\pm 5^\circ$ laminates in composite resizing, considerably greater generality is achieved in ASOP-3. It is now possible to accommodate laminates consisting of up to six layers, where a layer is defined as the aggregate of all laminae of a given material and fiber direction, and the fiber directions can be arbitrary. It is still necessary, however, that the strength of the laminate be "fiber-controlled" (Refer to Subsection 2.3). An option in the program permits the balancing of the numbers of laminae in specified pairs of layers.

In the analysis of composite structures to determine nodal deflections and internal loads, the full stiffness properties of the composite laminae, including the contribution of the matrix material, are taken into account. The internal loads are determined in this way for the whole laminate, which is then treated as a unit in the strength resizing process. This manner of resizing permits the introduction of desirable conservatisms, which are similar in nature to corresponding conservatisms applied in the resizing of the more restricted class of laminates in ASOP-2.

The first of these involves the neglect of the load-carrying capability of the matrix material. It is implemented by neglecting matrix stiffness in transforming internal loads (in the form of element stress resultants) to strains and in the subsequent transformation of these strains to layer stresses, which are needed for resizing. As the determination of laminate stiffness properties requires a prior knowledge of the laminate layup, it is necessary to use a convergent iterative process to design the layup for the given stress resultants, starting with the design for which the internal loads were determined in the analysis of the whole structure.

The other conservatism relates to the fact that, when all three components, N_x , N_y and N_{xy} , of the stress resultant acting on the laminate are non-zero, one or more of the layers may be less severely stressed than if N_x or N_y were zero. It is felt in some circles that, because of the dynamic nature of some of the critical loading conditions, the N_x and N_y components may not be acting simultaneously, and it may consequently be wise to include cases where N_x and N_y are separately set equal to zero as additional loading cases for laminate resizing. An option to apply such "cutoffs" is provided in ASOP-3.

An additional advantage of treating the entire laminate element as a unit in strength resizing is that it permits the introduction of a criterion for failure in microbuckling, a highly localized buckling of the composite fibers. Furthermore, it provides a proper framework for the future introduction of a capability of accommodating failure criteria for more general, or panel, buckling of such elements. Such a framework is not provided in programs that treat the layers of a laminate as distinct elements.

A significant advance has been made in ASOP-3 over earlier ASOP versions in the area of design optimization in the presence of deflection constraints. As in ASOP-2, a deflection-constraint resizing procedure, based on an optimality criterion involving gradients to a deflection-constraint surface, is used. This procedure has been improved in ASOP-3. Most importantly, it can now treat laminated composites, sizing each layer independently to satisfy the deflection constraint, unless layers are to be balanced, in which case the combined effect of such layers is taken into account. The interaction between stress and deflection constraints is taken into account in a manner which permits smooth and rapid convergence to a design that satisfies both constraints with near-minimum structural weight. Furthermore, the deflection

constraint is not limited to a translational displacement at a discrete node of the structural model. It can be represented as a linear combination of translational displacements, in specified degrees of freedom, at a number of given nodes. This permits the introduction of constraints on angular displacements, or on deformation shapes such as lifting-surface camber.

While deflection-constraint resizing is limited, at present, to a single constraint in a single-loading condition (although as many as twenty loading conditions can be applied in stress-constraint resizing), this limitation can be effectively circumvented in special cases. In particular, the case of a cantilever structure, such as a high-aspect-ratio wing or tail surface, with constraints on angle-of-twist at several spanwise stations, can be treated. This is done by means of successive submissions of the program. In the first submission, only the innermost constraint is applied. A second submission is then made, in which only the next outboard constraint is applied, with the gages of all members inboard of the innermost constraint location being kept fixed in deflection-constraint resizing at the values yielded by the first submission. In subsequent submissions, constraints are applied at successive locations, moving outboard, each time keeping all gages of members inboard of the last constrained station fixed at their latest values. This procedure cannot be expected to yield an exact result, but it should provide a design satisfying the constraints approximately.

Typical results are presented for two models. The first is a relatively coarse model of a swept wing, suitable for use in preliminary design. The second is a much more refined model of a bomber fin. In both cases, results are presented for a design based on the application of stress and minimum-gage constraints and another design based on the additional application of deflection constraints. In the case of the swept wing, twist constraints are applied at two spanwise stations, using the procedure outlined above.

Section 2

DESCRIPTION OF PROGRAM

2.1 INTRODUCTION

Various portions of the program are distinguished from one another on a functional basis. Prior to any resizing step, either for stress constraints or for a deflection constraint, it is necessary to analyze the structure, that is, to solve the equations relating nodal displacements to applied loads, and to determine the corresponding internal loads in individual elements. Accordingly, one portion of the program is concerned with this analysis operation. Another portion of the program takes the internal loads yielded by this analysis, and uses them in resizing the elements for stress constraints and constraints imposed by specified minimum and/or maximum gages. It should be noted that minimum gages usually represent limitations associated with practical construction, while the maximum gage is normally used as a means of fixing an element's gage by setting the minimum and maximum gages equal to one another. A third portion of the program takes the nodal deflections yielded by the analysis portion of the program, and uses them, in conjunction with other information, in resizing the elements for an imposed deflection constraint, taking cognizance also of the stress constraints and specified minimum or maximum gages.

When a deflection constraint is to be applied, two different phases, or "modes," in the redesign process are distinguished. In the "stress-constraint mode," which is the one that is executed first, a number of cycles of analysis and resizing for stress and minimum and/or maximum gage constraints are performed, until a convergence criterion is satisfied or the number of cycles has reached a specified maximum. The design should then be fully-stressed or nearly fully-stressed.

The "deflection-constraint mode" is then entered with that design, and deflection-constraint resizing and stress-constraint resizing are done sequentially within each cycle in that mode, with an analysis following each type of resizing. There are thus two analyses performed in each cycle in the deflection-constraint mode. This cycling in the deflection-constraint mode is continued, until a convergence criterion is satisfied or the number of cycles has reached a specified maximum.

2-2 STRUCTURAL ANALYSIS AND MODELING

The analysis portion of the program, in which displacements and internal loads are determined for given external loading, assumes linear elastic behavior of the structure and applies the matrix displacement method of finite-element structural analysis. This method is well-documented (References 5 and 6) and is based on the replacement of the actual structure by a mathematical model, or idealization, consisting of finite elements that are interconnected at discrete points or nodes. All external loads and displacement constraints are applied at these nodes.

The types of finite elements that can be accommodated in the program are listed in Table 1 and are described in some detail in Appendix A.

The type-2 beam element is generalized to provide for offset of the element nodes from the beam centroidal axis. This permits more realistic modeling of common components, such as fuselage frames, which have skin attached to one side. This feature is not applicable to the type-11 beam element, which is hinged at one end about one of the transverse axes.

The triangular membrane element is based on the assumption that the strain (hence stress) is uniform within the element, leading to full displacement compatibility between elements. The quadrilateral membrane element is constructed by assembling four of these triangular elements in a nonoverlapping arrangement, so that they have a common node at the intersection of the lines that connect the midpoints of pairs of opposite sides of the quadrilateral. Two versions of this element (types 5 and 8) are available: one in which all four nodes lie in a common plane, and the other in which this requirement is relaxed and a modest amount of warpage is permitted. The interior node is not loaded externally and any out-of-plane load that develops as a consequence of the warp is "beamed" to the corner nodes. The warped quadrilateral element is intended for use in idealizing lifting surface covers and fuselage skins that are only slightly warped.

The shear element (type 6) is a quadrilateral panel based on Garvey's assumptions (Reference 7). It is very useful in modeling rib and spar webs and, because some warpage is permitted, it can be used also for fuselage skins.

Table 1. LIST OF ELEMENTS IN ASOP-3

Element	Element No.	Description
Bar	1	Uniform cross section
Beam	2	Prismatic, offset
Beam	11	Prismatic, hinged at one end
Triangular Membrane	4	Constant-strain membrane
Quadrilateral Membrane	5	4 constant-strain triangles in the same plane
Quadrilateral Membrane	8	4 constant-strain triangles, warped
Quadrilateral Shear Panel	6	Planar or warped

Experience indicates that, for good results, quadrilateral elements should be as nearly rectangular as possible, and their aspect ratio should be kept below 2. Triangular elements should be avoided, if possible. If they must be used, they should be kept as nearly to an equilateral shape as possible. Grids should be refined in regions of high stress gradient and may be made coarser in regions of relatively low stress gradient. The idealization, in the transition between such regions, should preferably resemble an orthogonal curvilinear network.

In the modeling of structures constructed of filamentary composite materials, only membrane elements (types 4, 5 and 8) can, at present, be used in the program for such materials. Components, such as rib and spar webs, that carry primarily shear load, will normally have a layup consisting mostly of balanced $+45^\circ$ and -45° layers. The shear properties of such a composite can then be introduced into type-6 elements, treated as being homogeneous. For these elements, rounding to integral members of laminae will then have to be done as a separate operation following final redesign.

Composite elements can have any number of layers, up to a maximum of six, where a "layer" is defined as the aggregate of all laminae of a given composite material (for example, graphite/epoxy), and with fibers in a given direction. The fiber directions for the different layers can be arbitrary, except that, in aggregate, the layers should constitute a laminate whose strength is fiber-controlled (as defined

in Subsection 2.3.3). It should be noted that the laminae in a layer, as defined above, will not normally be contiguous in an actual laminate, but, in membrane action, their combined effect will be the same as if they were contiguous.

A composite material, as defined in the program for the purpose of data input, consists of the whole laminate, including number of layers, filament direction in each layer (relative to a reference direction which, as seen later, is the x_p -axis in the property axis system), and properties of the material in each layer. Starting and, if desired, minimum and maximum numbers of laminae in each layer are also specified.

Layers of composite elements are treated internally in the program as separate elements, except in resizing for stress constraints, where the interaction between them is taken into account, as discussed in Subsection 2.3.3. The layers of a composite element thus constitute a stack of elements that are connected only at the corner nodes. In the case of quadrilateral elements, which, as previously mentioned, consist of an assemblage of triangular elements with a common interior node, there may be some relative displacement of these interior nodes among layers in a laminate. This effect is believed to be small, however, and should not introduce any significant error.

In applying the matrix displacement method for the analysis of the structure, the required data is organized into data blocks as follows:

1. Nodal geometry
2. Boundary conditions
3. Applied loads
4. Material properties
5. Member data

The nodal geometry consists of the coordinates of the nodes in an orthogonal global axis system. The boundary conditions specify the nodes that are to be constrained against displacement and, for each constrained node, the coordinate directions in which this constraint is to be applied. The applied loads are specified in terms of the components of concentrated loads applied at given nodes. A maximum of twenty loading conditions may be applied in the analysis of the structure and its resizing to satisfy stress constraints. In the specification of material properties, three different classes of materials are distinguished: isotropic, orthotropic, and composite, with

composite materials defined as discussed above. If the elastic properties of a member are more generally anisotropic, they may still be introduced, but the necessary data must then be entered through member data cards. The members are the finite elements of the structural model, and the terms "member" and "finite element" or "element" are used here interchangeably. The member data specifies the element type, the material code, the nodes it connects, and other required geometrical data and program clues. In addition, material stiffness and strength data may be specified, if desired, with the member data rather than through the material property data, as explained in Subsection 4.2.6. In the case of composite members, the layer properties may be introduced on member data cards, instead of through the material properties input.

In the case of one-dimensional members or elements (bars and beams), a single system of orthogonal coordinate axes suffices. In the case of two-dimensional elements, however, at least two systems of orthogonal axes are used, as shown in Figure 1 for a quadrilateral element. One of these, named the "local element axes," x_ℓ, y_ℓ, z_ℓ , is oriented so that the origin is at node i , where the nodes are entered in the program in the order i, j, k, ℓ , and are arranged as shown in Figure 1. The x_ℓ -axis is along the edge i - j , positive toward j ; the y_ℓ -axis is positive toward the side on which the element lies; and the z_ℓ -axis completes a right-handed triad. While the direction i to j is shown as being counterclockwise around the element in Figure 1, it can just as well be clockwise, as long as node k is on a common edge with node i , and node ℓ with node j , and the axes are as defined above. In the case of planar elements, the local element axes are in the plane of the element, while in the case of warped quadrilateral elements, they are in a plane defined by a pair of straight lines joining the midpoints of opposite sides, as discussed in Appendix E.2.

The use of a second set of axes, the "property axes," x_p, y_p , is mandatory when the material is not isotropic. These axes, which are in the same plane as the x_ℓ and y_ℓ axes, are aligned with directions having significance in the definition of material properties. For example, in the case of orthotropic materials, they are the axes of symmetry in the material properties. For composite materials, they can be arbitrary, but are normally related in some simple manner to the fiber directions, as, for example, in the case of a $0^\circ/90^\circ/\pm 45^\circ$ laminate, where a natural choice for the x_p -axis would be the 0° direction. It is necessary to define property axes, even for

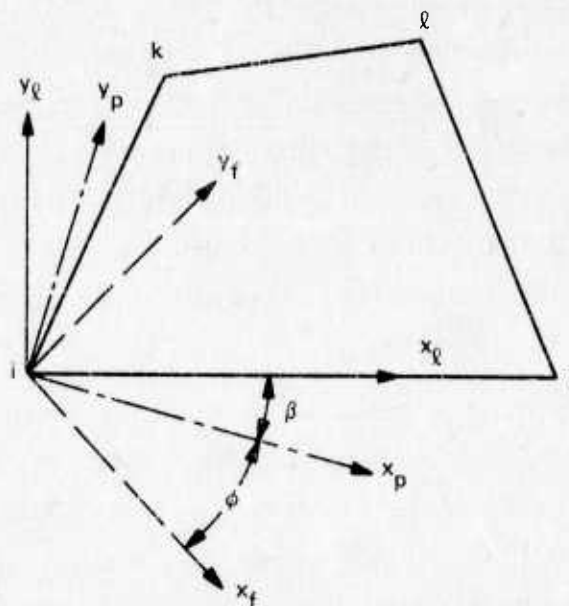


Figure 1. Member Axis Systems

isotropic elements, when the user desires that the stress and strain output not be referred to the local element axes. The property axes then define the axes to which the stresses and strains are to be referred.

The angle between the x_l and x_p axes is referred to as the " β -angle," and is positive as shown in Figure 1; that is, when a rotation from x_l to x_p is away from the element, rather than into the element. There are two ways of specifying it for each member. One is by entering it directly with the data for each member, which requires that it be precalculated for all the members - a task that can be formidable. The other is by separating the members into groups or "zones", the β -angles for all the members in each zone being calculated internally in the program, on the basis of a "reference direction" defined for that zone; the β -angle being the angle between the projection of the reference direction, in the plane of the member (or the i-j-k plane in the case of warped elements), and the x_l -axis. The relations used in that calculation are presented in Appendix C. Only the zone number need then be specified for each member, the reference direction being defined elsewhere for each zone.

In the case of composite members, there is a third set of orthogonal axes defined for each layer. These axes are the "fiber axes," x_f, y_f , which are in the same plane as the x_p and y_p axes (and also the x_ℓ and y_ℓ axes) and are aligned so that the x_f -axis is in the fiber direction. The angle between the x_f and x_p axes is the angle ϕ defined for each layer in the input for the composite material. It is positive in the same sense as β ; that is, when a rotation from x_p to x_f is away from the element, as shown in Figure 1.

While the various systems of axes have been defined above with reference to quadrilateral elements, the definitions remain the same for triangular elements.

Using the geometry, boundary condition, material property, and member data, the program calculates a stiffness matrix for each member, or each layer of a composite member. This matrix relates nodal forces to nodal displacements. After transformation from local element to global coordinates, these stiffness matrices are assembled, or "stacked," to form the stiffness matrix of the whole structure. This process involves a superposition of the element stiffness matrices as follows:

$$[K] = \sum_{i=1}^n \alpha_i [k_i] \quad (2.1)$$

where $[K]$ is the stiffness matrix of the whole structure, after the application of boundary conditions; $[k_i]$ is the stiffness matrix of an individual element, expanded to include all nodes of the structure (with appropriate boundary conditions applied to it) and normalized for a unit value of the design variable for that element; α_i is the design variable for that element (for example, the cross-sectional area of a bar or the thickness of a membrane element); and n is the total number of elements. In the case of beam elements, the radii of gyration of their cross section are assumed to be fixed, so that bending stiffness is directly proportional to cross-sectional area, which is the design variable. It should be noted that, in the case of composite elements, the matrix $[k_i]$ in Equation (2.1) is the stiffness matrix for an individual layer, per unit thickness of that layer, and α_i is the corresponding layer thickness.

The $[k_i]$ matrices are calculated only once and stored. In each redesign cycle, they are multiplied by the current values of α_i and reassembled to form a new $[K]$ matrix.

After formation of the structure stiffness matrix, the matrix equation:

$$[K][\Delta] = [P] \quad (2.2)$$

in which $[P]$ is the matrix of applied loads, is solved for the nodal displacements $[\Delta]$. The matrices $[P]$ and $[\Delta]$ have as many rows as there are degrees of freedom in the model, and as many columns as there are applied loading conditions. Each node has as many degrees of freedom as there are global coordinate directions in which it is free to displace.

Equation (2.2) is solved by a modified form of Cholesky's algorithm (See Appendix D.1). The displacements obtained in this manner are then substituted into the following matrix equation, which relates nodal forces in all the elements to the nodal displacements:

$$[q] = [S_1][\Delta]$$

where $[q]$ is a matrix containing the nodal forces in all the elements. The nodal forces in each element are given for a unit value of the design variable for that element and in the directions of the edges of the element (as shown in Appendix B). In the case of composite elements, the nodal forces are given for each layer of the element. The matrix $[S_1]$, referred to as the "member load matrix," is constructed by assembling the corresponding matrices for individual elements, or layers of composite elements, after performing a transformation that makes them compatible with displacements in global coordinates. It is formed only once and stored for use when required. The actual element nodal forces for each element, or each layer of a composite element, are obtained by multiplication by the current value of the element's design variable.

2.3 THE STRESS CONSTRAINT MODE

2.3.1 Basic Procedure and Element Stress Determination

The basic procedure for resizing based on allowable stresses and member gage limitations, such as minimum sheet gage for practical construction, is essentially the same as that in previous versions of ASOP. The stiffness matrix of each finite element in the structural model is assumed to be linearly related to a single design variable for that element (bar cross-sectional area, skin gage, etc.). In the case of the beam

elements, as discussed previously, it is assumed that the radii of gyration of the cross section remain constant, so that the moments of inertia of the cross section, and consequently the bending stiffness, are proportional to the cross-sectional area, which is the design variable in this case.

An initial design, specifying values of the design variables, is selected, and an analysis to determine nodal deflections and element stresses is carried out, for a given set of applied loading conditions. The state of stress in each element, for each loading condition, is then used in conjunction with a failure criterion to determine a "stress ratio". This provides a measure of the extent to which the stress constraint is satisfied or violated, and is discussed in detail in Subsections 2.3.2 and 2.3.3. It is equal to 1.0 if the failure criterion is exactly satisfied. The maximum value of the stress ratio, for all loading conditions for each element, is then used as a multiplying factor in resizing the design variable for that element. The procedure is considerably more complicated in the case of composite elements, as explained in Subsection 2.3.3.

This process of analysis and redesign is repeated cyclically, until a given number of cycles have been performed, or until a convergence criterion is satisfied. A converged design is referred to as a fully-stressed design; that is, one in which each element is stressed to the maximum allowable extent, in at least one loading condition, without being overstressed in any loading condition, or is at a minimum or maximum prescribed gage.

As discussed in Section 1, the nodal stress method for element stress determination, which is used in ASOP-1 and ASOP-2, has been found to have shortcomings, and it is not being used in ASOP-3. Instead, average stresses are determined for each element directly from the nodal forces acting on that element. In the case of the bar element, which is a uniform-strain element, the stress is simply the quotient of the nodal force (which is the uniform axial load in the bar) and the cross-sectional area of the bar. The triangular membrane element is similarly based on the assumption of uniform strain, and the average stress in it is simply the uniform stress associated with the uniform strain. The matrix transformation relation between average stress and the corner forces in that element is derived in Appendix E. It is based on a derivation in Reference 22.

The determination of average stress in the quadrilateral membrane elements (types 5 and 8) is not as straightforward, as they are constructed of four triangular elements, the strain being uniform in each of these triangles, but generally differing from one to the other. There are a variety of ways, necessarily approximate in nature, in which average stress can be defined in such an element. The definition chosen for use in ASOP-3 is one that has been developed at the Grumman Aerospace Corporation, for incorporation in its COMAP-ASTRAL structural analysis program, which is presently part of the RAVES system of compatible design programs. It is based on equilibrium considerations. A derivation of the applicable relations is presented in Appendix E, as transcribed from Reference 22. The average shear stress in the shear panel (element type 6) is determined in the same way.

In the beam elements, types 2 and 11, the axial load and corresponding stress are determined in the same way as in the bar element. Bending moments, however, are determined separately at each end of the element, and, because it is assumed that the element is loaded only at its ends, the maximum bending moment occurs at one of the ends. The shear force and torsional moment, both uniform along the element, are also determined.

2.3.2 Resizing Algorithm for Noncomposite Elements

The resizing of bar elements is particularly simple because of the uniaxial stress state in them, the stress ratio being the ratio of the actual stress to the allowable stress. In the resizing of beam elements, it is assumed that the element is loaded in bending primarily about only one of the two transverse axes: the z-axis. The bending moment about that axis is then determined at the two ends of the element, and corresponding extreme-fiber stresses are determined at the two ends, assuming the distance of the extreme fiber from the neutral axis to be equal to the radius of gyration of the cross section (which, in effect, assumes that the bending material is concentrated at the extreme fiber). The bending stresses are then combined with the stress due to the axial load, yielding four values of stress - two extreme-fiber stresses at each end of the element - and corresponding stress ratios are determined. The largest of these stress ratios is then selected for resizing purposes.

The biaxial stress state in membrane elements requires that a failure criterion, providing for the interaction between stress components, be used. In the case of isotropic materials, it is common to use the von Mises yield criterion, with

ultimate allowable stresses usually replacing the yield stresses, and that criterion is used in ASOP-3, in the following modified form:

$$\sqrt{\left(\frac{\sigma_x}{F_x}\right)^2 - \left(\frac{\sigma_x}{F_x} \frac{\sigma_y}{F_y}\right) + \left(\frac{\sigma_y}{F_y}\right)^2 + \left(\frac{\tau_{xy}}{F_s}\right)^2} = 1 \quad (2.3)$$

where F_x is the allowable tensile stress F_t or the allowable compressive stress F_c , depending upon the sign of σ_x , and similarly for F_y , while F_s is the allowable shear stress. F_t , F_c and F_s are always taken as positive quantities. The stress ratio in this case is the left-hand side of Equation (2.3).

In the case of orthotropic materials, the picture is considerably more blurred, and there is no universally accepted failure criterion. Two relatively simple criteria, having a somewhat rational basis, are Hill's generalization of the von Mises criterion and a criterion developed originally at the Forest Products Laboratory (References 8 and 9). In the absence of conclusive experimental evidence favoring either one of these criteria over the other, the latter has been selected for use in ASOP-3.

It is expressed in the form:

$$\sqrt{\left(\frac{\sigma_x}{F_x}\right)^2 - \left(\frac{\sigma_x}{F_x} \frac{\sigma_y}{F_y}\right) + \left(\frac{\sigma_y}{F_y}\right)^2 + \left(\frac{\tau_{xy}}{F_s}\right)^2} = 1 \quad (2.4)$$

$$\frac{\sigma_x}{F_x} = 1$$

$$\frac{\sigma_y}{F_y} = 1$$

where σ_x , σ_y and τ_{xy} are the stress components in the property axis system; F_x , F_y and F_s are the corresponding allowable stresses in the absence of the other components; and failure is presumed to occur when any one of the three relations is satisfied. F_x and F_y are tensile or compressive allowable stresses, as appropriate, and are always taken as positive quantities.

In applying this criterion, the left-hand sides of all three relations are evaluated, and the largest of the three becomes the governing stress ratio, which is then used as a multiplying factor in resizing, as described in Subsection 2.3.1.

The failure criterion expressed by Equation (2.4) is shown graphically as an envelope, in Figure 2, for the case, $\tau_{xy} = 0$. The first of the equations is represented by the ellipse, and the remaining two equations effectively apply cutoffs in the first and third quadrants. It can be seen that the ellipse, in Figure 2, also represents Equation (2.3), which is the failure criterion for isotropic materials. If desired, the cutoffs shown in Figure 2 can be applied to isotropic materials as well, by suitable adjustment of the input data, as explained in Subsection 4.2.5.

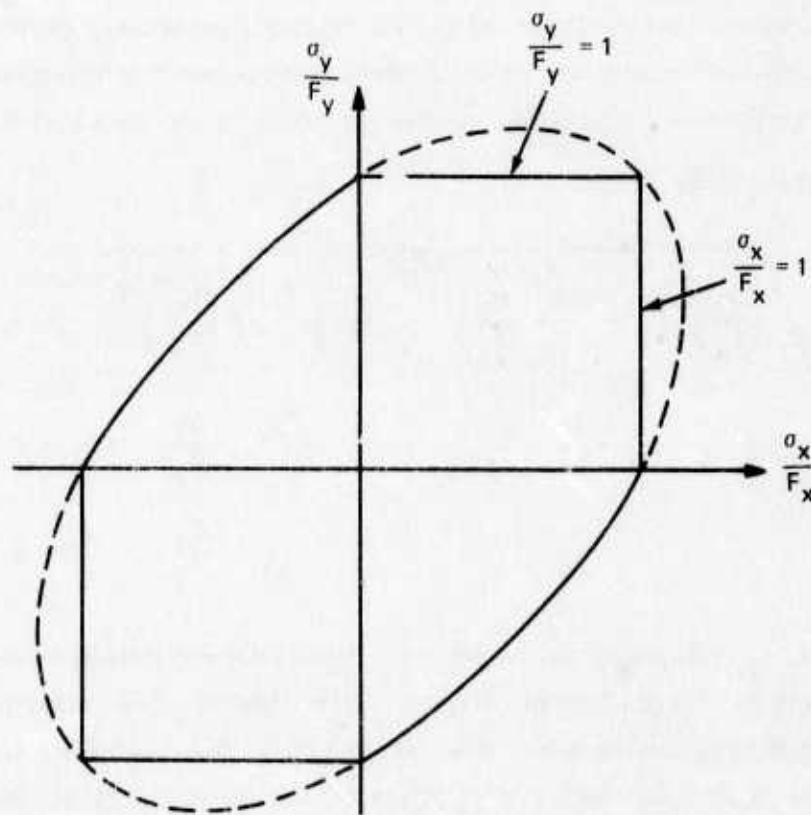


Figure 2. Failure Criterion for Orthotropic Materials in Biaxial Stress

The new element gages obtained in this way are checked against the minimum gages specified in the input data, and are revised accordingly, if necessary. If the stress resizing is being done as part of a cycle in the deflection-constraint mode, as discussed in Subsection 2.3.2, a similar check is made against the minimum gages established by deflection-constraint resizing, and necessary revisions are made. A check is then made against the maximum gages specified in the input data, and necessary revisions are made.

While composite elements can (it is expected that they usually will) be resized by the comprehensive procedure described in the following section, a cruder approach can be used, in which they are treated as orthotropic elements and resized by the procedure described above. In that case, it is necessary that the layup be such that the relative numbers of laminae, in the various fiber directions, are maintained constant so that appropriate allowable stresses can be selected. It will be seen that the cutoffs described in the following section bear some similarity to those shown in Figure 2.

2.3.3 Resizing Algorithm for Composite Elements

The criteria governing the failure of composites are more complex than those governing the failure of noncomposite materials. In consequence, the algorithm for composite element resizing is necessarily more complex, requiring that the laminate be treated as a unit, so that interaction between layers may be properly taken into account. Furthermore, because of limited operational experience with composite materials, it is desirable to make certain conservative assumptions concerning their strength behavior.

For example, local cracking or crazing in the matrix material may greatly reduce its effectiveness as a load-carrying agent, even though it continues to serve its central purpose as a binding agent. For that reason, the assumption is made here that all the load is carried by the filaments.* This assumption is applicable only to so-called filament-controlled composites: those in which the layup is such that

*Note that, as stated in Section 1, this assumption is made only in the resizing process. In the analysis of the whole structure, to determine nodal displacements and internal loads, the full stiffness properties of the composite elements, including the contribution of the matrix material, are used.

filament directions are sufficient in number and distribution that any component of the laminate stress resultant can be resisted by filaments alone. It would not be a tenable assumption in the case of matrix-controlled composites: those in which the load-carrying capability of the matrix is relied upon. The ASOP-3 program does not, at present, accommodate composites of this latter type, although it can be further developed to do so.

Another conservatism, one that may be applied as an option in the program, relates to interaction between laminate layers that is somewhat akin to the effect of hydrostatic stress in metals or other nominally homogeneous materials. It is known, for example, that, if components N_x and N_y of the laminate stress resultant are present and are of the same sign, some layers will be less severely stressed than if either N_x or N_y were absent. Some designers feel that it is unconservative to take advantage of this fact and base the design on the simultaneous presence of both components, particularly since the prediction of applied loads is hardly an exact science, and the dynamic nature of some loading conditions suggests the possibility that different components of the internal loading may not be applied simultaneously. The program option referred to as the "cutoff option," makes it possible to provide additional stress checks, with N_x and N_y set successively to zero, and to use the results as additional information in the resizing process.

In addition to filament failure in tension or block compression, the possibility of failure in the so-called "microbuckling" mode may be taken into account. In that mode, there is a highly-localized buckling of the filaments because of their own low bending stiffness and the limited shear stiffness of the matrix material, which is relied upon to resist such buckling (Reference 10). Theoretically, it has been found that the allowable stress, G_z , in this mode, should be equal to $G_m/(1 - V_f)$, where G_m is the shear stiffness of the matrix material, and V_f is the volume fraction of fibers (Reference 10); however, experimental evidence indicates that it is better to use a value based on experimental data (Reference 11). If experimental data is not available, the theoretical value may be multiplied by an empirical coefficient which, at least for the case of boron/epoxy, can be given the value 0.63 (Reference 12).

To take proper account of interaction between layers, the resizing algorithm for composite elements must treat the entire laminate as a unit and apply a convergent iterative procedure. This procedure, which is summarized in Figure 3 in the form of a flow chart, consists of the following steps:

- (a) The nodal forces for each layer of the element, initially determined on a per-unit-thickness basis in the analysis of the structure, are multiplied by the actual thickness of the layer prior to resizing. They are then summed for all layers, to yield nodal forces for the whole laminate, for all applied loading conditions.
- (b) The laminate nodal forces are transformed to average stress resultants, for all applied loading conditions, using the procedure of Appendix E. 1 for triangular elements and that of Appendix E. 2 for quadrilateral elements.
- (c) Using the components of the stress resultants determined in Step (b), the principal stress resultants are computed for each loading condition, as shown in Appendix F. 2. The largest negative (compressive) value of the principal stress resultant, for all loading conditions, is selected for use in applying the microbuckling failure criterion.
- (d) The stiffness matrix relating stresses and strains is determined for each layer, neglecting the matrix stiffness, and with reference to axes aligned with the filament direction. Each of these stiffness matrices is then transformed to the property axes for the element, as described in Appendix F. 1.
- (e) Using the current laminate layup and the values of the allowable stress, G_z , in the microbuckling failure mode for all layers of the laminate, an effective value of G_z is determined as an average of the individual layer values, weighted on the basis of layer thickness, as shown in Appendix F. 2. It should be noted that this calculation is necessary only when a hybrid laminate is used.
- (f) The layer stiffness matrices determined in Step (d) are multiplied by the current values of the corresponding layer thicknesses and summed for all layers, to yield a matrix relating the stress resultants for the whole laminate to the strains, as shown in Appendix F. 1.

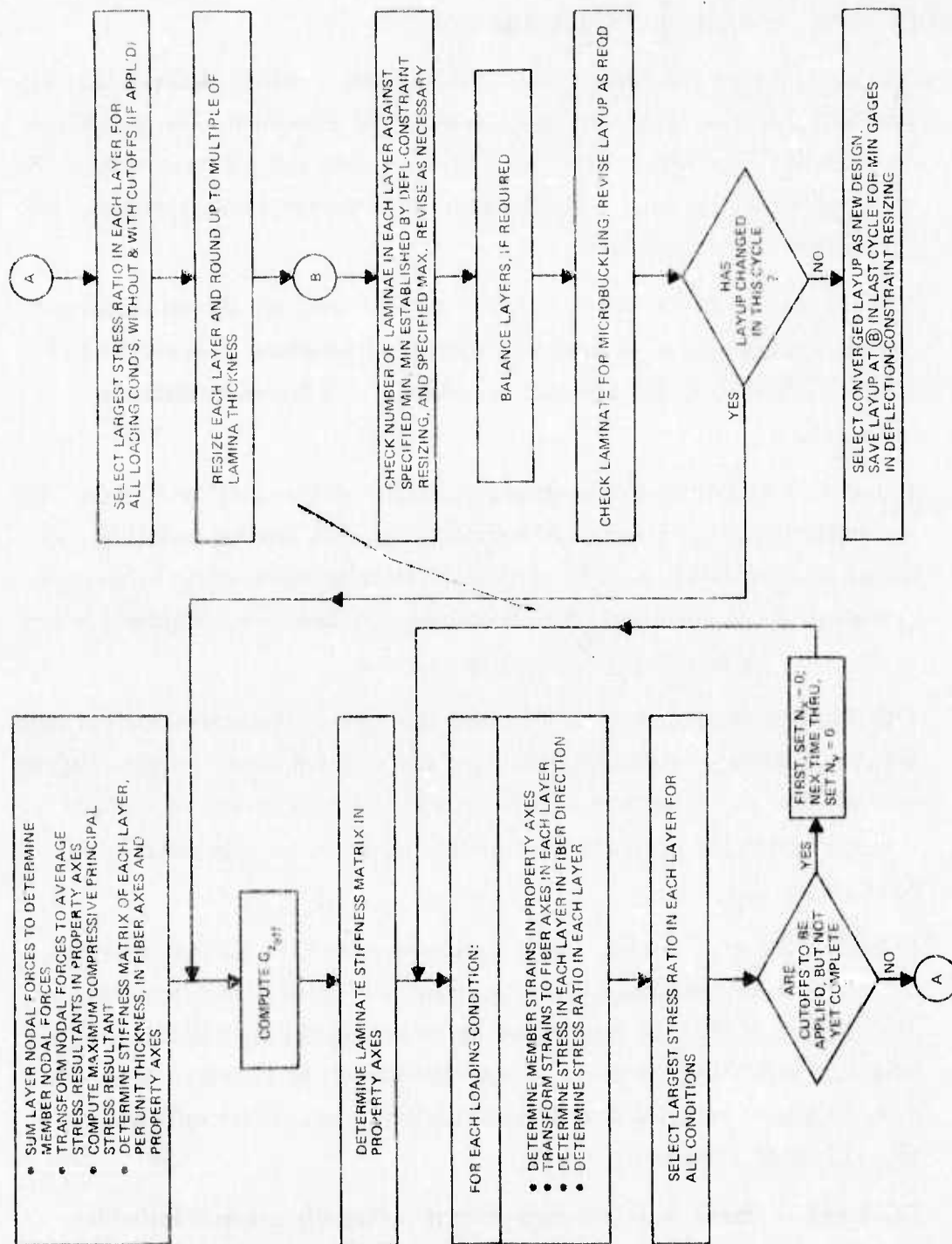


Figure 3. Cyclic Process for Stress-Constraint Resizing of Composite Elements

- (g) The stiffness matrix determined in Step (f) is used in conjunction with the stress resultants determined in Step (b) to solve for the strain state in the laminate corresponding to each loading condition. This requires the solution of Equation (F-1) in Appendix F.1.
- (h) The strain states determined in Step (g) are transformed to filament coordinates in each layer, and are then used in conjunction with the layer stiffness matrix determined in Step (d), in the same coordinate system, to determine filament stress, as shown in Appendix F.1. Because of the neglect of matrix stiffness, this stress is a uniaxial stress along the filaments.
- (i) The filament stresses determined in Step (h) are divided by the allowable filament tensile or compressive stress, whichever is appropriate, to yield a stress ratio for each layer, in each loading condition. The largest value of the stress ratio for each layer, in all loading conditions, is then selected for resizing purposes.
- (j) If the cutoff option is exercised, Steps (g), (h), and (i) are repeated, first with N_x set equal to zero, and then with N_y set equal to zero (after restoration of N_x to its original value).
- (k) The largest value of the stress ratio obtained for each layer in Steps (i) and (j) is used as a multiplying factor to resize the layer thickness. The layer thickness obtained is then divided by the thickness of an individual lamina, and the result rounded up to the nearest integer, to determine the number of laminae in each layer.
- (l) The layup obtained in Step (k) is checked against the minimum number of laminae specified for each layer in the input data, and is revised accordingly, if necessary. If the stress resizing is being done as part of a cycle in the deflection-constraint mode, as discussed in Subsection 2.3.2, a similar check is made against the minimum number of laminae established by deflection-constraint resizing, and necessary revisions are made. A check is then made against the maximum number of laminae specified for each layer in the input data, and necessary revisions are made.

- (m) If pairs of layers are to be balanced, as, for example, the balancing of the $+45^\circ$ and -45° layers in a $0^\circ/90^\circ/\pm 45^\circ$ layup, the number of laminae in the layers of such a pair are compared and, if they are not already equal, the smaller is set equal to the larger.
- (n) Using the new layup, as it exists at this point, a total laminate thickness is computed and is used in conjunction with the largest compressive stress resultant, as determined in Step (c), to determine a maximum principal compressive stress for the laminate. This stress is divided by the effective value of G_z , determined in Step (e), to yield a stress ratio for microbuckling, which is multiplied by the total laminate thickness just computed, to determine a laminate thickness required to satisfy the microbuckling failure criterion, as shown in Appendix F.2. If this new thickness does not exceed the thickness associated with the layup determined by the other criteria, it is disregarded, and the microbuckling mode does not govern the design. Otherwise, the difference in thickness is made up by adding laminae to layer number 1, rounding up to the nearest whole number of additional laminae. This procedure requires that a prior decision be made as to which layer is to be increased, to satisfy the microbuckling criterion, and that the layer selected be designated as layer number 1.
- (o) If, as a result of Steps (e) through (n), the laminate layup has changed, the new layup becomes the "current" layup, and corresponding current-layer thicknesses are determined. Steps (e) through (n) are then repeated cyclically, until the layup does not change in two successive cycles, or until a specified maximum number of cycles have been performed. The converged layup is then taken as the new design.

2.3.4 Resizing of Compression Panels

The limited capability, introduced into ASOP-2, for the resizing of bars and shear or compression panels subject to buckling failure, has been retained in ASOP-3. It involves the introduction of "stability tables" relating allowable stresses to internal loading. These tables may be generated by the user or a small subsidiary program, developed for the purpose, may be used. This capability is limited to noncomposite elements. The reader is referred to Reference 3 for further details.

2.4 THE DEFLECTION CONSTRAINT MODE

2.4.1 Resizing Algorithm

As in ASOP-2, resizing for deflection constraints is accomplished in ASOP-3 by the application of an optimality criterion. It has been shown (References 14 and 15) and is demonstrated in Appendix G.1 that, for the case of a single deflection constraint and in the absence of other constraints, a minimum weight is achieved when the partial derivative of the constrained deflection with respect to element weight has the same value for all elements. That is:

$$\frac{\partial \delta}{\partial w_i} = K \quad (i = 1, 2, \dots, n) \quad (2.5)$$

where δ is the deflection to be constrained, w_i is the weight of the i^{th} element (of a total of n elements), and K is a constant.

When, as in most practical designs, there are strength constraints and minimum or maximum gage constraints in addition to the deflection constraint, the uniform-derivative criterion, expressed in Equation (2.5), can be applied to the set of all elements not governed by these other constraints, the corresponding element weights being referred to as the "active" variables. In that case, the criterion is less rigorously applicable, but should still give a design of nearly minimum weight, as discussed in Appendix G.1

The minimum-weight design cannot be arrived at directly. It is necessary to employ an iterative process, which has been found to converge rapidly in practical cases. The iterative process used in ASOP-3 is similar, but not identical, to that used in ASOP-2 (Reference 3) and in a number of other references, including References 16, 17 and 18. In the details of its application, it resembles most closely the procedure used in the FASTOP program for optimization to satisfy a constraint on flutter velocity (Reference 15).

Starting with a nonoptimum design, which may or may not satisfy the prescribed deflection constraint, the following recursion relation, derived in Appendix G.2, is applied in successive cycles:

$$w_{i_{\text{new}}} = w_{i_{\text{old}}} \sqrt{\frac{\left(\frac{\partial \delta}{\partial w_i}\right)_{\text{old}}}{\left(\frac{\partial \delta}{\partial w}\right)_{\text{target}}}} \quad (2.6)$$

where $w_{i_{\text{old}}}$ is the weight of the i th element, prior to resizing in the current cycle; $w_{i_{\text{new}}}$ is the weight of the i th element, following resizing in the current cycle; $(\partial \delta / \partial w_i)_{\text{old}}$ is the partial derivative of the constrained deflection with respect to w_i , computed for the design existing prior to resizing in the current cycle; and $(\partial \delta / \partial w)_{\text{target}}$ is a quantity given the name "target derivative" and is defined below. The basis for Equation (2.6) is discussed in Appendix G.

At the optimum design, the target derivative will be the constant K in Equation (2.5), and the derivatives $\partial \delta / \partial w_i$ will all be equal to it. However, prior to convergence to an optimum design, the derivatives $\partial \delta / \partial w_i$ will differ from each other in value, and, in fact, may differ in sign. Depending upon the sign of the target derivative, some of these derivatives may then yield a negative value for the quantity under the radical in Equation (2.6). The corresponding elements will then have to be excluded when Equation (2.6) is applied. This is discussed further in Subsection 2.4.2.

As the value of K in Equation (2.5) is not known until the optimum design is achieved, it is necessary to find a value for the target derivative that, when introduced into Equation (2.6), will yield a design that satisfies the constraint, at least approximately. This is done by a trial procedure, in which a value of the target derivative is sought that will yield a design satisfying the relation:

$$\delta_{\text{desired}} = \delta_{\text{old}} + \sum_{i=1}^n \frac{1}{2} \left[\left(\frac{\partial \delta}{\partial w_i} \right)_{\text{old}} + \left(\frac{\partial \delta}{\partial w} \right)_{\text{target}} \right] (w_{i_{\text{new}}} - w_{i_{\text{old}}}) \quad (2.7)$$

where δ_{old} is the value of the constrained deflection prior to resizing; δ_{desired} is the desired value of the constrained deflection in a resizing step; and the summation is over all elements of the model, including those to which Equation (2.6) is not applied.

Equation (2.7) is seen to provide a second-order approximation (in the Taylor Series sense) to the desired value of the deflection. An exact value could have been obtained by a trial procedure, in which a structural analysis is performed for the design corresponding to each trial value of the target derivative, to determine the deflection subject to constraint. However, as this operation would have to be performed a number of times, for successive trial values of the target derivative, and is expensive computationally, it is highly advantageous and, in practice, satisfactory to use Equation (2.7) instead.

It was stated above that Equation (2.6) is applied to that group of elements with derivatives, $\partial\delta/\partial w_i$, that are of the same sign as the target derivative. The determination of that sign is now considered. It is established upon entry into the deflection-constraint mode and depends upon whether the constraint value of the subject deflection exceeds, or is less than, its current value (for the design existing upon entry into the deflection-constraint mode). If the constraint value exceeds the current value algebraically, and the constraint is either (1) an equality constraint or (2) an inequality constraint that has been violated, then an increase in deflection is desired, and the proper derivative sign is that which is associated with an increase in deflection resulting from an increase in element weight, that is, a positive sign. For those elements with negative derivatives, a reduction in element weight will move the deflection in the desired direction, and the design variables for these elements can be permitted to decrease, to the extent permitted by other constraints. In the reverse situation, where the constraint value is less than the current value, Equation (2.6) is applied to those elements with negative derivatives. Where the constraint is an inequality constraint and is not violated by the design existing at exit from the stress-constraint mode, no further resizing is necessary.

The sign of the target derivative will, by definition, be the same as that of the derivatives $(\partial\delta/\partial w_i)_{old}$ introduced into Equation (2.6). It remains to find a value of the target derivative that will satisfy Equation (2.7). This is done by taking, as an initial trial value, upon entry into the deflection-constraint mode, a value equal to 80% of the average of all the derivatives having the proper sign, determined as explained above. (In subsequent redesign cycles of the iterative redesign process, as explained below, the starting value of the target derivative is the last value computed in the preceding cycle.) The target derivative is then incremented until a value is achieved that satisfies Equation (2.7), within a tolerance specified by the user.

The value of δ_{desired} in Equation (2.7) is not necessarily the constraint value. It may be advantageous, in some situations, to move from the initial value of the subject deflection to the vicinity of the constraint value in a series of shorter steps, rather than in a single step. Equation (2.7) then provides a closer approximation in each step. Furthermore, when the constraint boundary in the design space is reached by this procedure, it may be at a point considerably closer to the optimum design point. Accordingly, the program provides an option that permits the change from the initial value of the deflection to the constraint value to be made in a number of approximately equal increments, that number being selected by the user.

The deflection that is subject to constraint may be generalized in the sense that it may be represented as a linear combination of nodal displacements in specified degrees of freedom. Thus, for example, an angular displacement constraint may be treated by representing it as the difference between the translational displacements of two specified points, divided by the distance between them. The two points specified need not be at nodes; their displacements can be obtained by interpolation between nodal displacements. Similarly, a given amount of camber of a lifting surface, at a given spanwise station, can be specified as a constraint, by a similar representation as a linear combination of nodal displacements.

When composite elements are included in the model, each layer of such elements is treated internally in the program as a separate element. Accordingly, Equation (2.6) is applied to individual layers, with the deflection derivative being computed for each layer with respect to that layer's weight. However, the derivatives of layers that are to be balanced are modified as explained in the following section. In the trial process of finding a value of the target derivative that satisfies Equation (2.7), the layer thickness is not rounded. Rounding is done only after that process is completed, when the layer thickness is rounded up or down to the nearest multiple of the lamina thickness. By rounding both up and down, the effect of rounding on the constrained deflection can be minimized.

The evaluation of the deflection derivatives is carried out in the same way as in ASOP-2 (Reference 3), and is explained in Appendix G.3.

2.4.2 Resizing Algorithm for Interacting Deflection, Stress, and Minimum/Maximum Gage Constraints

A detailed exposition of the deflection-constraint algorithm, as it is applied in the presence of stress constraints and specified member gage limitations, is now presented.

As stated above, the deflection-constraint mode is entered only if (1) the deflection constraint is an equality constraint ($\delta = \delta_{\text{desired}}$), or (2) it is an inequality constraint ($\delta \leq \delta_{\text{desired}}$ or $\delta \geq \delta_{\text{desired}}$) that is violated by the design existing at the end of the stress-constraint mode. (It should be noted that the subject deflection δ and its desired value δ_{desired} are algebraic quantities, not absolute values.) It is desirable that the design existing at entry to the deflection-constraint mode should be very nearly a fully-stressed design.

If the constraint is an inequality constraint that is violated by the design existing at entry to the deflection-constraint mode, it is treated as an equality constraint in the deflection-constraint mode. It is shown in Appendix G.4 that this approach should yield a design that is near optimum for the inequality constraint.

Prior to entry into the iterative redesign process, a determination is made of the algebraic sign of the derivatives to be introduced into Equation (2.6), as discussed in Subsection 2.4.1. If the current value of the deflection subject to constraint is smaller (algebraically) than the constraint value, positive derivatives are taken. If the current value of the deflection subject to constraint is larger (algebraically) than the constraint value, negative derivatives are taken. Once this determination is made, it remains unchanged throughout the remainder of the procedure, for reasons that will be discussed later.

The resizing procedure in the deflection-constraint mode is now outlined as a sequence of steps. These steps can be broken down into two distinct groups. One group comprises the major steps in an iterative cycle in the deflection-constraint mode, with alternating deflection-constraint and stress-constraint resizing steps and intervening structural analyses. These steps are summarized in flow-chart form in Figure 4. The other group, consisting of steps (c) through (g), constitutes an inner loop for deflection-constraint resizing, in which successive trial values of the target

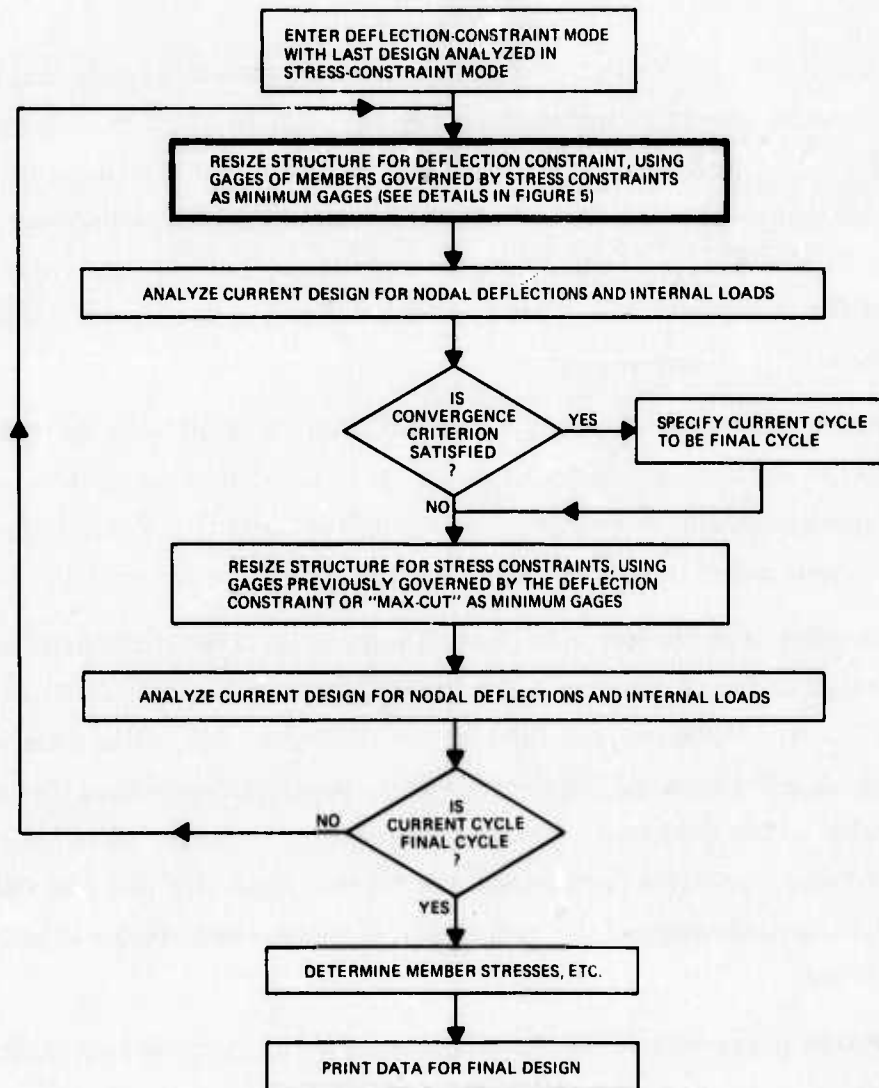


Figure 4. Iteration Cycles in Deflection-Constraint Mode

derivative are introduced, until an acceptable value is obtained. The steps in this inner loop are shown in flow-chart form in Figure 5. The whole sequence of steps, in both the inner and outer loops, is now presented as follows:

- (a) The derivatives $\partial \delta / \partial w_i$ are determined for all elements (or layers of composite elements) for the current design (which, in the first cycle, is the last design analyzed in the stress-constraint mode). The procedure described in Appendix G is used in this determination. In the case of balanced layers of composite elements, the derivatives obtained for these layers are averaged and the average value is used in place of the actual values. This has the effect of introducing the combined influence on the deflection of equal changes in the thickness of such layers.
- (b) An initial trial value of the target derivative is selected. In the first cycle in the deflection-constraint mode, it is equal to 80% of the average value of all the deflection derivatives with the proper sign for the application of Equation (2.6). In subsequent cycles, it is the final value of the target derivative in the immediately preceding cycle.
- (c) Using the current trial value of the target derivative, those elements (or layers of composite elements) with derivatives of the proper sign are resized by application of Equation (2.6), rewritten here in the following form:

$$\alpha_{i_{\text{new}}} = \alpha_{i_{\text{old}}} \sqrt{\frac{\left(\frac{\partial \delta}{\partial w_i}\right)_{\text{old}}}{\left(\frac{\partial \delta}{\partial w}\right)_{\text{target}}}} \quad (2.8)$$

where α_i is the element gage and can replace w_i , as w_i is directly proportional to α_i . Each $\alpha_{i_{\text{new}}}$ computed in this way is compared with the "max cut" value, $k\alpha_{i_{\text{old}}}$, and, if smaller, is made equal to that value. The "max cut" value, $k\alpha_{i_{\text{old}}}$ (where k is a coefficient having a value between 0 and 1, specified by the user) determines the maximum amount by which any element gage (or composite layer thickness) will be permitted to decrease in one cycle.

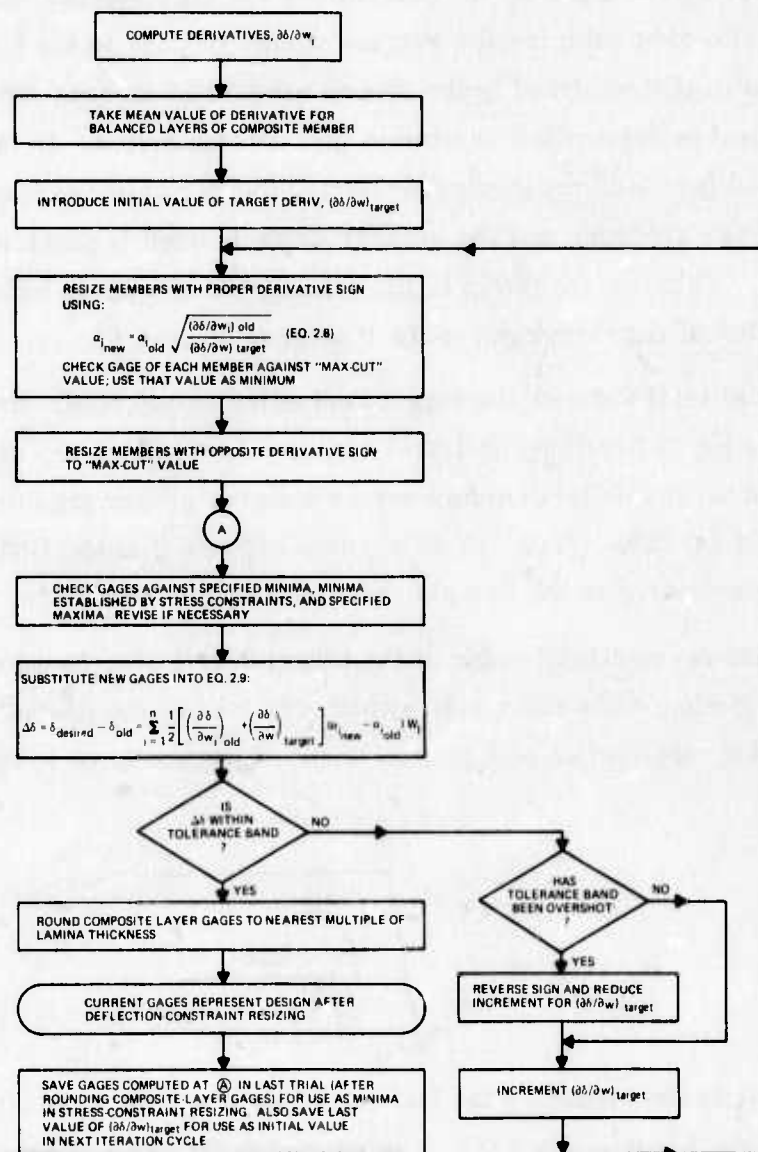


Figure 5. Algorithm for Deflection-Constraint Resizing

- (d) The gages of all elements that are not resized by Equation (2.8) are set at their "max cut" value, $k\alpha_{i \text{ old}}$.
- (e) Each element gage (or composite layer thickness), $\alpha_{i \text{ new}}$, yielded by Steps (c) and (d), is checked against the larger of the following, which is then applied as a minimum gage:
 - (1) The element gage (or composite layer thickness) determined by application of the stress constraint, before application of any other constraint, in the latest stress resizing. In the case of composite elements, it is the layer thickness determined the last time through Step k in the cyclic process described in Subsection 2.3.3.
 - (2) The element minimum gage (or composite layer minimum gage) as specified in the input data.
- (f) Each $\alpha_{i \text{ new}}$ yielded by Step (e) is compared with the element maximum gage (or composite layer maximum gage), as specified in the input data, and is revised as necessary.
- (g) The new values of α_i (unrounded in the case of composite layers), determined in Steps (c) through (f) for all elements, are introduced into the right-hand side of Equation (2.7), rewritten as follows:

$$\delta_{\text{desired}} = \delta_{\text{old}} + \sum_{i=1}^n \frac{1}{2} \left[\left(\frac{\partial \delta}{\partial w_i} \right)_{\text{old}} + \left(\frac{\partial \delta}{\partial w} \right)_{\text{target}} \right] (\alpha_{i \text{ new}} - \alpha_{i \text{ old}}) W_i \quad (2.9)$$

where W_i is the weight of the i th element, per unit value of its gage; and subscript "old" applies to values existing prior to deflection-constraint resizing in the current iteration cycle and remaining fixed in the trials for successive values of the target derivative. As explained previously, δ_{desired} is not necessarily equal to the constraint value, $\delta_{\text{constraint}}$. If it is desired to divide the approach to the constraint value in a series of N steps, the value of δ_{desired} will be given by:

$$\begin{aligned} \delta_{\text{desired}} &= \delta_{\text{old}} + \frac{\delta_{\text{constraint}} - \delta_{\text{old}}}{N - j + 1} \quad (j = 1, 2, \dots, N-1) \\ &= \delta_{\text{constraint}} \quad (j \geq N) \end{aligned} \quad (2.10)$$

where j is the number of the cycle in the deflection-constraint mode. If Equation (2.9) is not satisfied, within a tolerance specified as described in Subsection 4.2.7, the target derivative is, at first, incremented by an amount equal to 0.2 times its initial value. This increment is either positive or negative, depending upon the direction in which it is necessary to move to achieve satisfaction of Equation (2.9). It is applied in successive trials, until the tolerance band for δ_{desired} is either entered or overshoot. If that band is overshoot, the increment is reduced, by another multiplication by the factor 0.2, and is reversed in sign. Such reduction and reversal in sign is done every time the tolerance band is overshoot, in either direction. In any case, if the tolerance band is not entered in any trial, Step (c) is then reentered with the new value of the target derivative, but with the member gages α_{old} remaining unchanged.

- (h) When Equation (2.9) is satisfied within the specified tolerance, deflection-constraint resizing in the current iteration cycle is deemed to be complete, except for rounding the gages of composite element layers to the nearest multiple of the lamina thickness (which involves both rounding up and rounding down).
- (i) The gages of those elements resized in the latest pass through Steps (c) and (d), but before application of other constraints, are saved. In the case of composite element layers, the saved gages are rounded, as in Step (h). These saved gages are applied as minimum gages in the next resizing for strength.
- (j) Using the design yielded by Step (h), the structure stiffness matrix is stacked, and a solution for nodal deflections and element nodal forces is carried out.
- (k) The deflection subject to constraint is evaluated, and a convergence test is applied. If this deflection is within a specified tolerance, and was within that tolerance also at the corresponding point in the immediately preceding cycle, and if the total structure weight has not changed by more than a specified amount between those two points, the current cycle becomes the final cycle.

- (l) Using the element nodal forces obtained in Step (j), the stress-constraint resizing algorithm, as described in Subsections 2.3.2 and 2.3.3, is applied. In this process, the element gages saved in Step (i) are used as minimum gages, along with the minimum gages specified in the input data.
- (m) Using the design yielded by Step (l), another structural analysis is performed, as in Step (j).
- (n) If the convergence test applied in Step (k) is not satisfied, Step (a) is reentered, using the design yielded by Step (l) and the results obtained in Step (m).
- (o) If the convergence test applied in Step (k) is satisfied, the design yielded by Step (l) is the final design, and the results of Step (m) are used to determine element stresses, strains, etc.

The alternate application of deflection and stress constraints, with the gages determined by one being used as minimum gages in the other, as described above, converges to a design in which there are two classes of elements (or layers in the case of composites). One class comprises elements that are fully stressed or are at minimum or maximum specified gage. The other class comprises elements that are governed by the deflection constraint. In this latter class, the derivatives of deflection with respect to element (or layer) weight all have nearly uniform values. Departures from uniformity are due to lack of convergence or, in the case of composite element layers, to rounding to an integral number of laminae. Under these circumstances, it can be expected that the design will be close to optimum, at least in a local, if not in a global, sense.

An explanation of why the sign of the deflection derivatives, for introduction into Equation (2.8), is kept unchanged throughout the iteration process, is now provided. As long as movement from the initial value of the constrained deflection (upon entry into the deflection-constraint mode) toward the constraint value is in the same direction, it is clear that this sign should not be changed. However, what about the situation where the constraint value is overshoot, and movement in the reverse direction becomes necessary? If the sign were to be reversed, all those elements previously resized by the deflection constraint would be suddenly relieved of such constraint, and their gages could drop to values determined by other constraints. Under these circumstances, large changes could be expected to result from a need for minor adjustments,

as the amount of overshoot would normally be small. These large changes could be expected to preclude satisfactory convergence to an optimum design. Maintaining the same sign keeps these adjustments essentially within the same group of elements that have previously been governed by the deflection constraint.

It should be noted that an option in the program permits the designation of selected members as noncandidates for deflection-constraint resizing, although they are still subject to the application of stress constraints in each cycle in the deflection-constraint mode, unless their gages are being explicitly fixed. This option is useful, for example, when designing structures having fixed-gage honeycomb core substructure that is idealized using shear panels. Another useful application is described in Subsection 2.4.3.

2.4.3 Treatment of Multiple Deflection Constraints as a Succession of Single-Constraint Problems

Although only a single deflection constraint, in a single loading condition, can be treated in one submission of the program, as discussed above, it is possible to treat special cases of multiple deflection constraints by making multiple submissions of the program. The special cases are those in which the constraints can be ordered, so that the first constraint can be satisfied, after which a portion of the structure can be frozen in design to prevent further change in the corresponding deflection; then the second constraint can be satisfied by redesign of the remaining structure, after which a portion of that structure can be frozen in design, to prevent further change in the corresponding deflection; and so on, until the last constraint is satisfied. Clearly, cantilever structures, particularly slender ones, with deflection constraints, such as angles of twist, applied at two or more stations along the span, fit this situation to some degree of approximation.

The program must be submitted for execution as many times as there are constraints to be satisfied. In each submission, the design variables are initialized at the final values they had in the preceding submission, and, in the case of those design variables that are nominally to be frozen, their newly initialized values are also their minimum values. The word "nominally" is used because, while these design variables are removed from candidacy for deflection-constraint resizing, they are not truly frozen, as it is still necessary to apply stress constraints to them if overstress is not to occur. This may have the effect of further altering the

deflections already set at their constraint values, but this effect can generally be expected to be small.

2.5 FLEXIBLE SUPPORTS

A feature of the ASOP-2 program is the capability of replacing rigid supports with elastic constraints that may be cross-coupled. It is useful in simulating the redundant supports represented by wing-fuselage connections and the constraints imposed on wing or tail surfaces by flexible control surfaces. This capability has been retained in ASOP-3. It is described in more detail in Reference 3.

Section 3

APPLICATIONS TO REPRESENTATIVE PROBLEMS

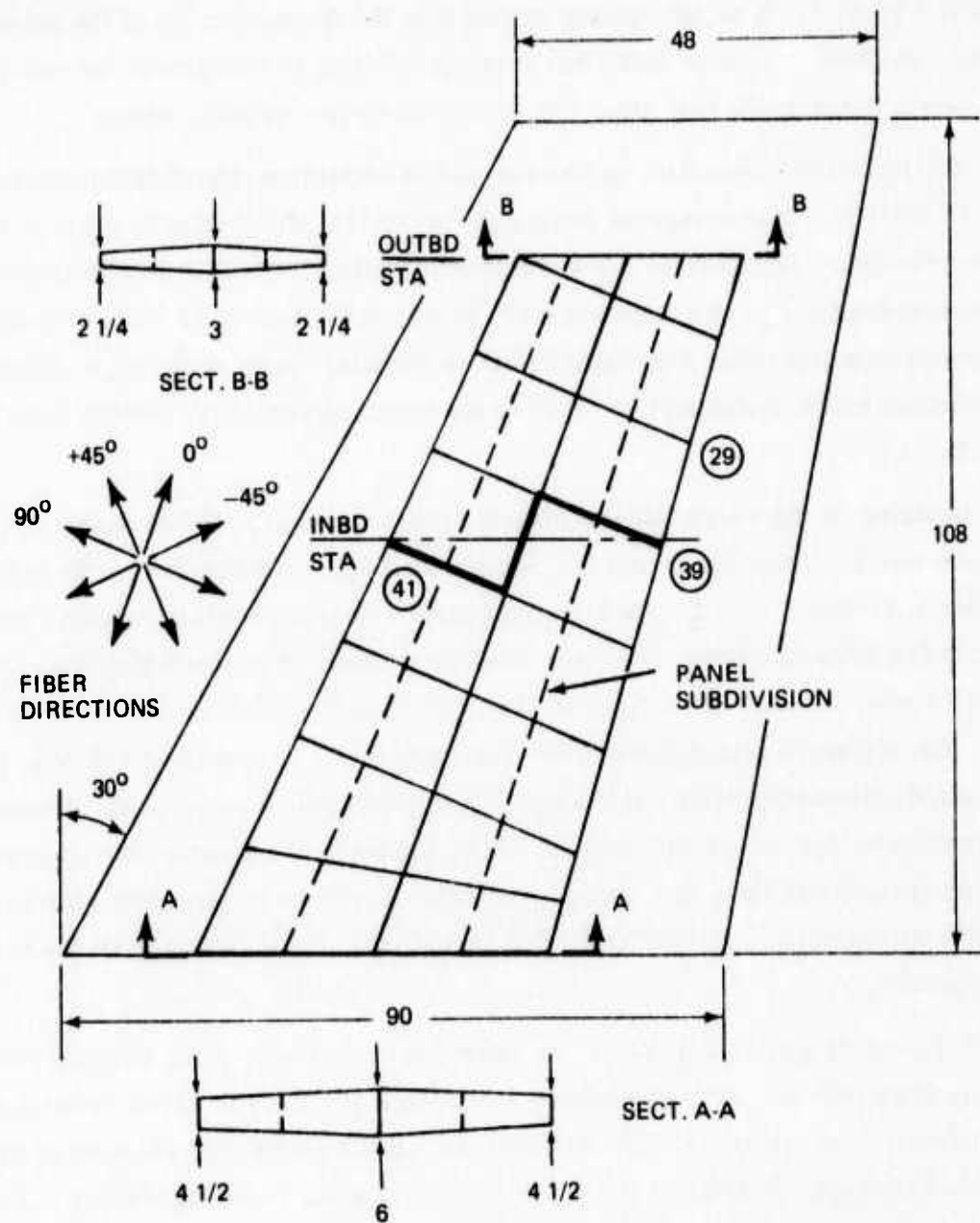
3.1 INTRODUCTION

The ASOP-3 program has been applied to two different structures, to demonstrate its capability for both stress-constraint resizing and deflection-constraint resizing. The two models representing these structures are of two quite different levels of detail, one being relatively coarse and serving primarily to demonstrate the resizing capability of the program, and the other being a much more refined model demonstrating, in addition, the program's capability of handling large problems requiring the performance of certain operations in segments rather than on the whole structure at one time. The two models and the results obtained for them are described in the following sections.

3.2 INTERMEDIATE-COMPLEXITY WING

The cantilevered wing model shown in planform in Figure 6 was chosen for study as an illustration of the application of the program in the preliminary design of a lifting surface. The primary structure of this "Intermediate-Complexity Wing" is a symmetric two-cell box beam having aluminum substructure and graphite/epoxy cover skins with fiber directions as shown in the figure. The solid lines (Figure 6) indicate the locations of shear webs. The substructure is modeled with conventional shear-panel elements and posts (which are the only bar elements in the model); the cover skins are composite four-layer elements which are permitted to have unbalanced $\pm 45^\circ$ layers. The model has 88 nodes and 158 members, and is built-in at the root.

Two applied loading conditions were generated by using simplified pressure distributions representative of a subsonic, forward-center-of-pressure loading and a supersonic, near-uniform-pressure loading. The structure was sized for these conditions in the stress-constraint mode without exercising the "cutoff" option. Satisfactory convergence was achieved in five cycles in the stress-constraint mode. The resulting design was then examined from the point-of-view of streamwise-twist distribution along the wing's span for the subsonic condition -- the twist angle being based simply on the difference in vertical displacements between the forward and aft



NOTE: ALL DIMENSIONS IN INCHES
EXCEPT WHERE OTHERWISE
NOTED

Figure 6. Aerodynamic Planform and Primary Structural Arrangement of Intermediate-Complexity Wing

wing spars along a streamwise chord. This twist distribution is shown by the upper curve in Figure 7. It is interesting to note that the forward c. p. of the subsonic loading distribution causes sufficient nose-up twisting to overpower the usual nose-down twisting (washout) that generally occurs in swept metallic wings.

To illustrate a potential application of the deflection-constraint resizing capability of ASOP-3, it was decided to attempt to "tailor" the design to achieve a prescribed streamwise-twist distribution for the subsonic loading condition that would offer improved aerodynamic performance through increased lift-to-drag ratio. Twist angles at two wing stations were then established as targets, these being -2.0° (washout) at a selected inboard station and -2.5° at the most-outboard rib station (see Figures 6 and 7).

Resizing in the deflection-constraint mode was accomplished in two stages. The approach was to divide the structure into two regions, as indicated by the bold separating line in Figure 6. In the first resizing stage, only the composite cover skin elements in the inboard region were permitted to be resized in the deflection part of a resizing cycle, to meet the inboard station twist-angle requirement. In the second stage, only outboard region cover skin elements were allowed to be resized, to achieve the desired outboard-station twist angle. In both stages, however, all elements were permitted to be resized if they were strength critical. This two-stage approach was based on the concept that, for high-aspect-ratio cantilevered surfaces, the resizing of elements outboard of a particular station should have little influence on the deflections at that station.

The first stage of resizing in the deflection-constraint mode started with the fully-stressed design. Convergence to the desired twist angle at the inboard station was achieved in seven steps, with the overall resulting twist distribution as shown by the dashed curve in Figure 7. Figure 8 summarizes the resizing history in this mode, in terms of inboard station twist angle versus total structural weight, after the strength-resizing part of each cycle.

In the second stage of deflection-constraint resizing, all starting gages were taken as those of the final design in the previous run. For elements in the inboard region, these starting gages were also treated as minimums, to prevent removal of material that was previously introduced to meet the inboard-station twist requirement. Convergence to the desired outboard-station twist angle required only two cycles,

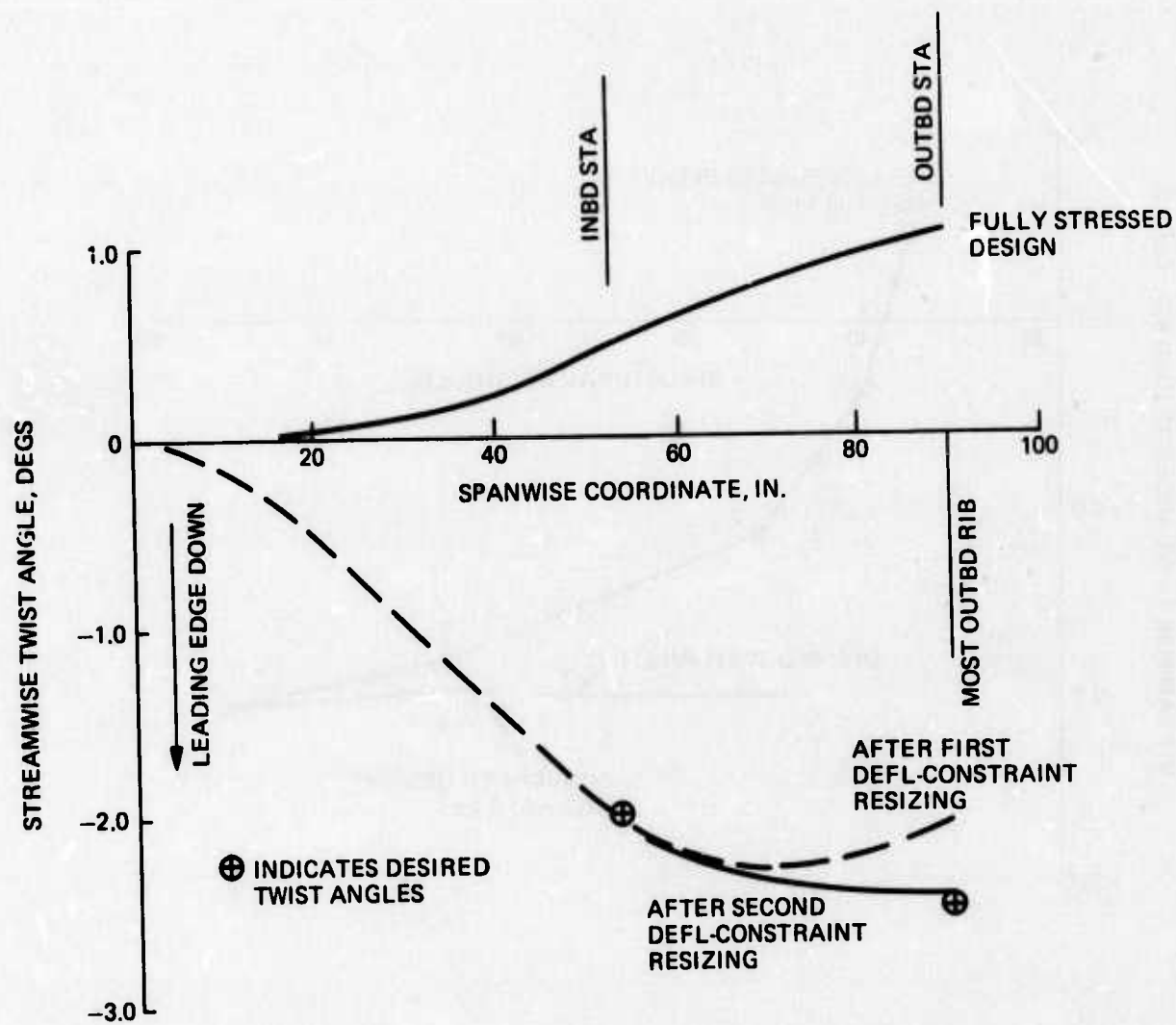


Figure 7. Intermediate-Complexity Wing -- Streamwise Twist Distribution at Various Stages of Resizing

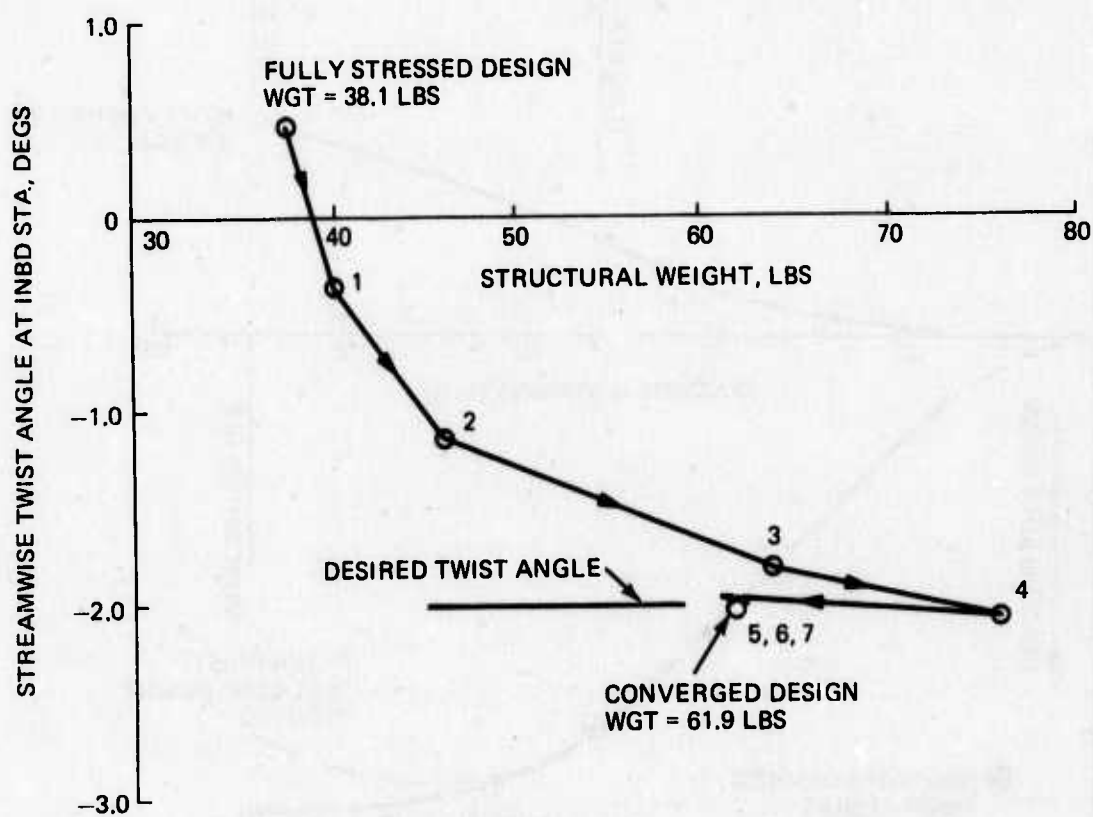


Figure 8. Intermediate-Complexity Wing — Resizing History from Fully-Stressed Design to Deflection-Constrained Design for Inboard Station

with only a very small additional weight increase. The final twist distribution after this second deflection-constraint resizing is shown in Figure 7, and a summary of results for all stages of resizing is presented in Table 2. It should be pointed out that the small differences between the target and accepted twist angles are due mainly to limits imposed by the practical requirement for rounding layers to integral numbers of laminae. Figure 9 displays the cover skin layer arrangement, for the initial fully-stressed design and the final combined strength and deflection constrained design. Figure 10 presents substructure gages for both designs.

TABLE 2. SUMMARY OF RESULTS FOR INTERMEDIATE-COMPLEXITY WING

Constraint Mode	Cycles To Convergence	(1) θ_{Inbd}	(2) θ_{Outbd}	Structural Weight
Stress	5	+0.47°	+1.06°	38.1 lb
First Deflection	7	-2.03	-2.05	61.9
Second Deflection	2	-2.04	-2.43	62.3

(1) Desired Value = - 2.00° (Leading Edge Down)

(2) Desired Value = - 2.50° (Leading Edge Down)

3.3 BOMBER FIN

The second model studied is that of a bomber fin, shown in planform in Figure 11. This model was supplied by the Air Force Flight Dynamics Laboratory and is a derivative of one that was used in the early design stages of an actual structure.

The support points of the model are shown as heavy dots in Figure 11. The forward support point is a single point on the structure's plane of symmetry. The remaining support points occur as pairs, symmetrically located with respect to the plane of symmetry. Shear webs and cover bar elements are used to simulate spars along all of the spanwise grid lines shown in Figure 11, and to simulate ribs at the root, tip and several intermediate locations. Posts are present at all of the grid points. All of these elements are fixed in gage and are of isotropic material, with the Young's modulus of aluminum. The same is true of the cover membrane elements in the two spanwise bays closest to the leading edge, as well as in the extended root structure. The remaining cover elements are of graphite/epoxy composite with fiber

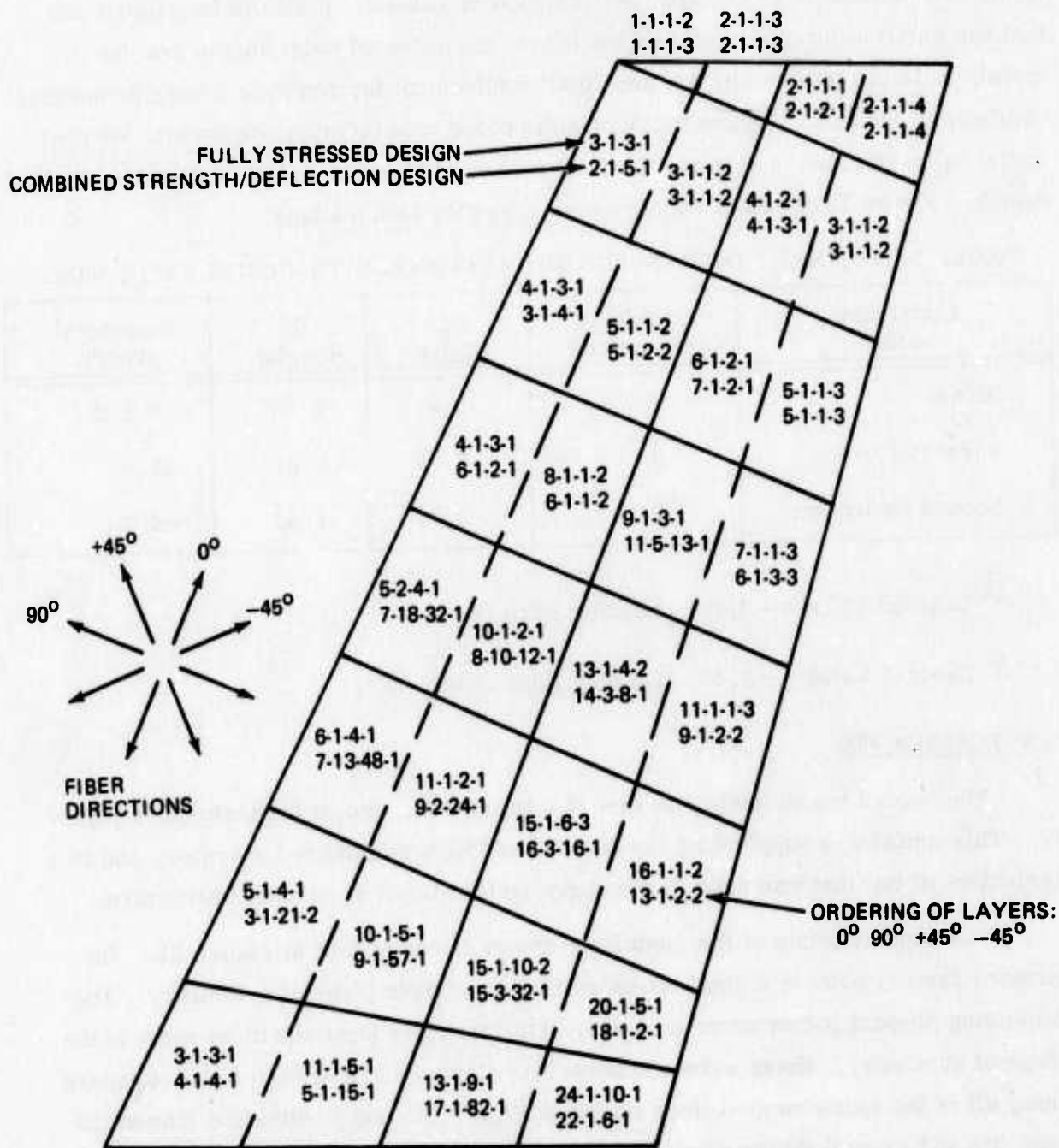


Figure 9. Cover-Skin Layups for Intermediate-Complexity Wing

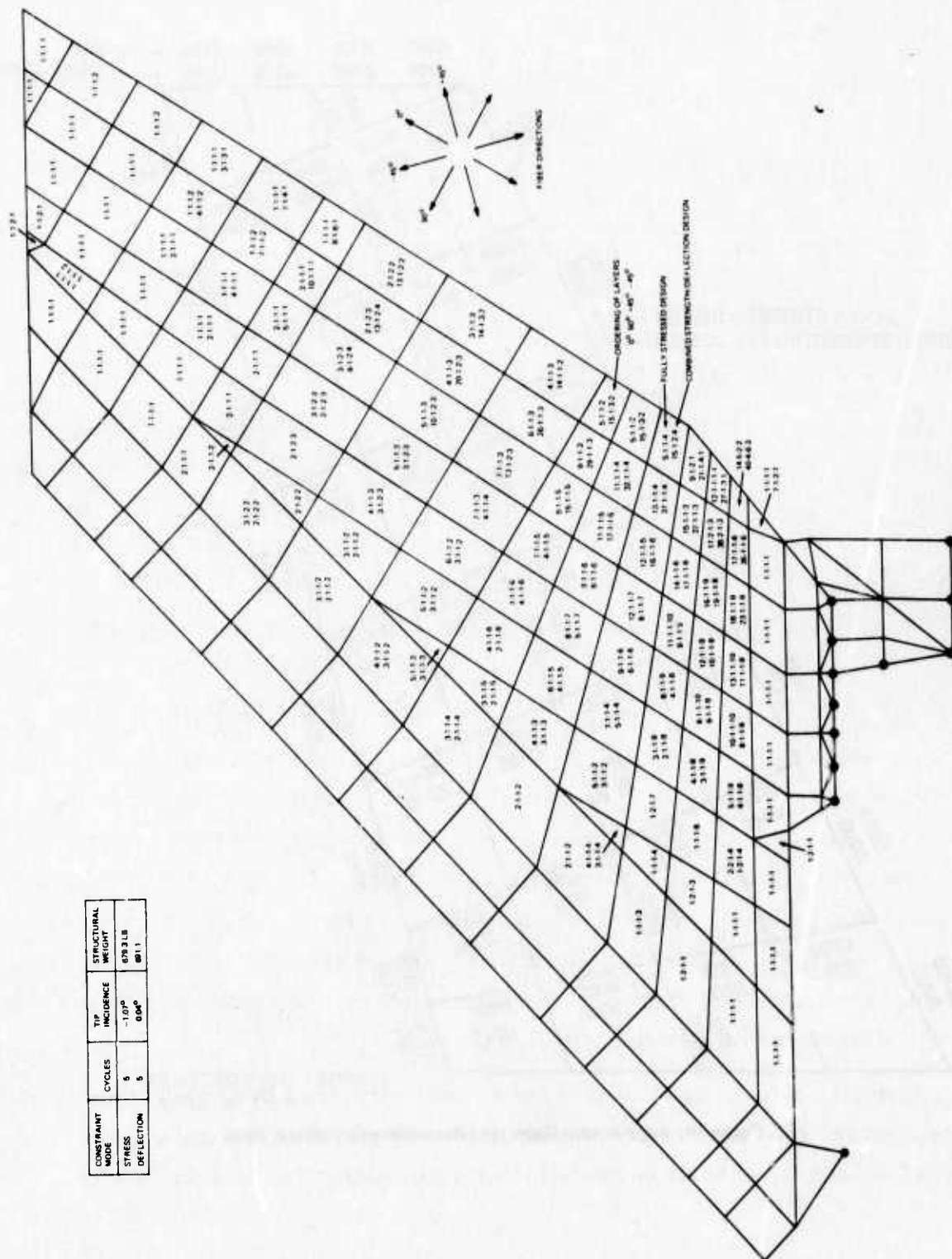


Figure 11. Bomber Fin Model and Cover-Skin Layouts

directions as shown in Figure 11, the $+45^\circ$ and -45° layers being allowed to be unbalanced. These composite elements are the only ones that are adjustable in the design. In the chordwise bay, just outboard of the extended root structure, there are aluminum membrane cover elements of fixed gage superimposed on the adjustable composite elements. The model contains 375 modes and 1293 members.

Five cycles of resizing were performed in the stress-constraint mode and an additional five cycles in the deflection-constraint mode. The stress-constraint resizing was based on eight applied loading conditions corresponding to a wide variety of flight conditions. The load distribution in each of these conditions was made precisely antisymmetric about the plane of symmetry of the fin, to insure that a symmetric design would be obtained. It should be noted that an alternate way of obtaining a symmetric design, without a restriction to antisymmetric loading, would be to introduce the loading conditions as matched pairs in which each condition is the mirror image of the other.

The deflection constraint applied was on the angular displacement of the tip chord relative to the root. It was applied in a loading condition in which the strength-governed design experiences a washout of 1.07° at the tip. The constraint was applied to eliminate this washout, so that the fin would be fully effective in this design condition; that is, it would have the same aerodynamic characteristics as a rigid structure. The cover layups corresponding to the designs existing at the end of the stress-constraint mode and at the end of the deflection-constraint mode are shown in Figure 11. Where there was no change from the former to the latter, only a single layup is shown.

It is seen from the tabular insert (Figure 11) that satisfaction of the deflection constraint required a total weight increase of only 1.7 percent. The reason for the smallness of this increase, as a percentage of the total weight, is twofold. First, the increase in composite thickness in the three spanwise bays closest to the trailing edge is partially offset by decreases in thickness in the more forward bays. Second, the composite cover material represents a relatively small part of the total weight of this structure. The increase in weight, as a percentage of the composite cover weight at the end of the stress-constraint mode, was 27.6 percent.

It should be noted that, while five cycles were performed in the deflection-constraint mode, good convergence was obtained after only four cycles, with the vicinity of the constraint value of the deflection being approached in a single step.

Section 4

USER INFORMATION

4.1 GENERAL DESCRIPTION AND LIMITATIONS

The ASOP-3 program, written in FORTRAN IV language, was developed on the IBM 370/168 computer. Versions of the program are available for execution on both IBM and CDC systems.

It should be noted that in the IBM version, the solution of Equation (2.2), yielding nodal displacements in the analysis of the structure, is performed using single-precision arithmetic. This has been found, in the case of structures for which the stiffness matrix is ill-conditioned, to provide insufficient accuracy. Accordingly, users of the IBM version are cautioned that this situation can arise. If it is suspected, one possible check is to determine whether the applied loads are in equilibrium with the support reactions. The components of the resultant forces and moments associated with the applied loads are provided in an output table (Subsection 4.3.2), and the support reactions may be determined from data in the table of cap forces (Refer to Subsection 4.3.5.) The much greater precision with which CDC systems perform arithmetic operations essentially precludes this problem, except in extreme cases.

Limitations on models that can be accommodated by the program are:

- (a) Maximum number of loading conditions = 20
- (b) Maximum number of nodes = 1000
- (c) Maximum number of degrees of freedom = 6000
- (d) Maximum number of members = 3000, with a further limitation for deflection-constraint resizing, as discussed below.

There are, additionally, limitations on the ordering of the nodes and members. While there is no absolute requirement with regard to node numbering and ordering, large savings in computing time can be achieved by properly numbering and ordering the nodes, to minimize the bandwidth of the structure's stiffness matrix. The bandwidth is determined by how far apart all pairs of connected nodes are in number. Accordingly, the node numbering should minimize this separation.

In the case of members, there is a firm limitation on ordering that arises because of the calculation of cap forces, which are summations of member nodal forces in the directions of grid lines at each node. These nodal forces are summed in blocks and stored, block by block. Consequently, members that have common nodes should not be very widely separated in the member sequence. A rough rule of thumb is that such members should not be separated by more than 100 intervening members. It should be noted that the sequence referred to here is the sequence in which the members appear in the input data, and not the numerical order of member numbers. The member numbers can be arbitrary, except that they cannot exceed four digits.

Deflection-constraint resizing is limited to a single constraint in a single loading condition, but the constraint can be generalized to a linear combination of translational displacements at specified nodes and in specified coordinate directions. The maximum number of degrees of freedom entering into such a linear combination has been set at 100. In addition to the limitation in the total number of members to 3000, there is a further limitation in the case of deflection-constraint resizing to 6000 submembers that are candidates for such resizing. In noncomposite members, there is no distinction between a member and a submember. In composite members, however, each layer is considered to be a submember. This restriction can be alleviated considerably by removing some members from candidacy for deflection-constraint resizing, as provided in the input. Noncandidates for deflection-constraint resizing can be those members that can be expected to have little or no influence on the deflection subject to constraint, or they can be members that are to be fixed in gage. The designation of such noncandidate members is useful also in applications where multiple deflection constraints are to be applied by means of successive submittals of the ASOP-3 program, as discussed in Subsections 2.4.3 and 3.2.

4.2 CARD INPUT AND ILLUSTRATIONS OF DATA PREPARATION

4.2.1 General Construction of Input Deck

The structure of the input deck is shown in Figure 12. Detailed information on the composition and format of individual segments of this deck is provided in the following sections.

The composition of the Job Control Language (JCL) deck depends upon the computing system being used. For a specific computing system (either IBM or CDC), it is available in two different versions. One version is for use when all output data is to

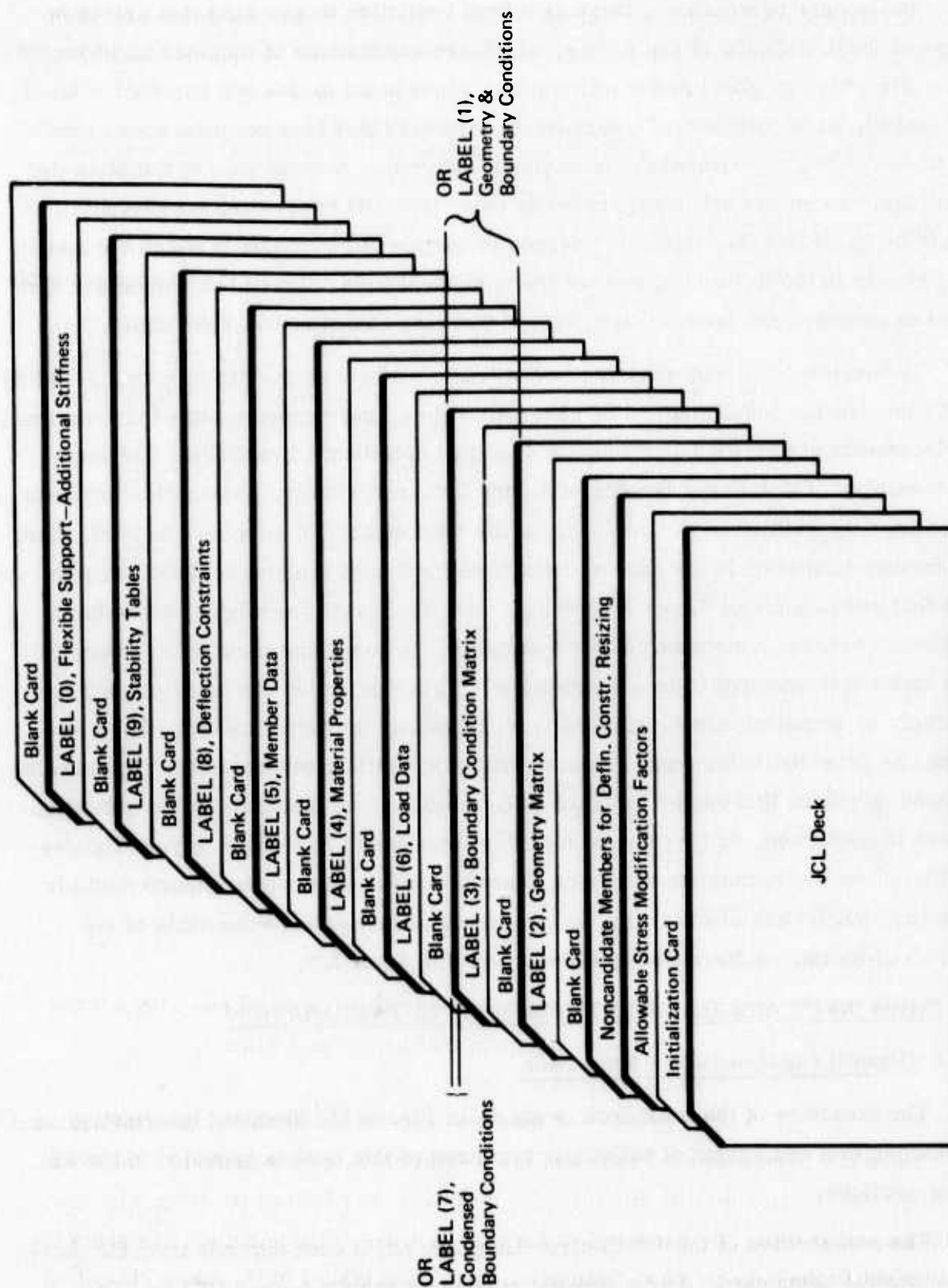


Figure 12. Construction of Input Deck

be printed out online, that is, as a single entity. The other version is for use when some of the output data is to be stored on a tape, available for offline or separate printout, if desired. This is discussed further in Subsection 4.3.5.

Following the JCL deck, an "initialization card," providing various controls, and cards for allowable stress modification factors and for the designation of members as noncandidates for deflection-constraint resizing are placed.

Following these cards, blocks of data describing the structural optimization problem to be solved are placed. These data blocks are categorized by classes. Ten classes are defined, but not all are used in any problem. They are listed as follows, in the order in which they would normally appear in the data deck rather than in numerical order:

- (1) Geometry and boundary conditions
- (2) Geometry only
- (3) Boundary conditions only
- (7) Condensed boundary conditions
- (6) Applied loads
- (4) Material properties
- (5) Member data
- (8) Deflection-constraint data
- (9) Stability tables
- (0) Flexible supports and additional stiffness data

All of the above listed data sets begin with a LABEL card and end with a blank card. The LABEL card has the following form that begins in column 6:

LABEL(i), NAMEA, NAMEB

where $i = 0, 1, 2, \dots, 9$ is the number of the data class as listed above, and NAMEA and NAMEB are any alpha-numeric names of up to eight characters selected by the user, either or both of which may be absent, except in the case of Class (6) data, where a special format is used, as discussed in Subsection 4.2.4. These names are generally used to identify the data sets and distinguish them from others in the same class that

may be used in other computer runs. No blank spaces should be left on the LABEL card between the fifth column and the first comma.

It should be noted that the sequence of the data sets through the LABEL (5) data set must be maintained as shown in Figure 12. The sequence of the LABEL (8), (9) and (0) data sets is immaterial.

4.2.2 Initialization Card, Allowable Stress Modification Factors, and Noncandidate Members for Deflection-Constraint Resizing.

Prior to the reading of any large blocks of data describing the structure and its loading, a small group of cards are introduced to provide certain controls on the operation of the program. The first of these cards, referred to as the "initialization card", contains data controlling the number of iteration cycles executed in the stress-constraint mode and a series of clues permitting the user to exercise options in the program. Its format is shown in Figure 13, and its content consists of the following:

Columns 1-2 - An I2 field containing the maximum number of iteration cycles to be executed in the stress-constraint mode. This number is never exceeded, even if the convergence criterion has not been satisfied. If this field is left blank or a zero is introduced, the program will perform a structural analysis of the design submitted. If a deflection constraint is not to be applied, or if an inequality deflection constraint is applied but is found not to be violated by the design submitted, the results of the structural analysis will be printed out, including nodal deflections, member stresses, etc. No resizing of members will take place.

Columns 3-8 - An F6.0 field containing the tolerance on the maximum stress ratio in all members. This tolerance is a parameter in the convergence criterion in the stress-constraint mode. If the maximum stress ratio is equal to 1.0 plus or minus this tolerance in two successive cycles, and if the change in total structure weight between these cycles does not exceed the value specified in the next field, no further iteration cycles are executed in the stress-constraint mode. If this field is left blank, the number of iteration cycle specified in the first field will be executed.

Columns 9-14 - An F6.0 field containing the maximum permissible change in total structure weight, in two successive cycles, in the stress-constraint mode, as discussed above in connection with the convergence criterion. If this field is left blank, the number of iteration cycles specified in the first field will be executed.

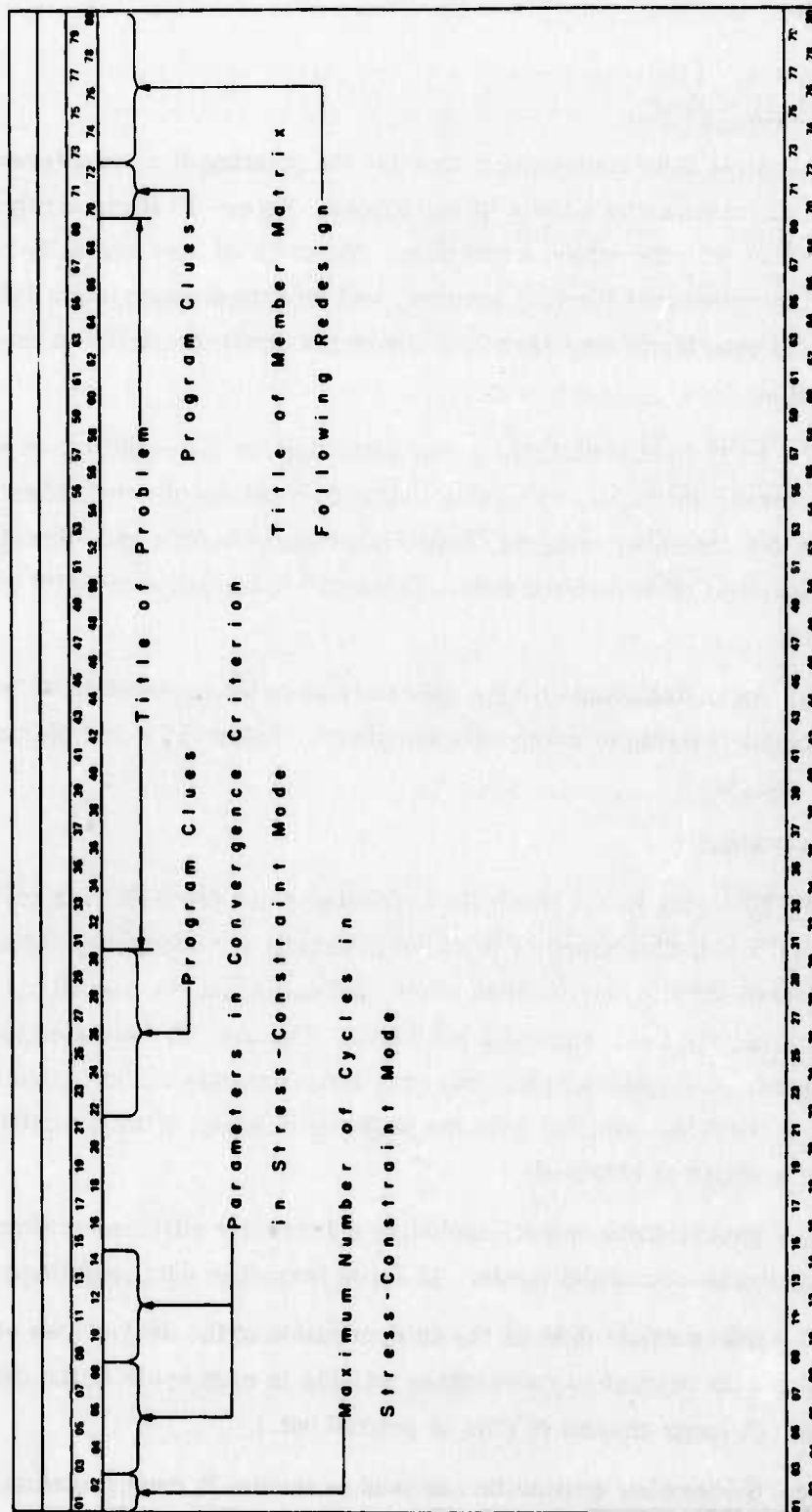


Figure 13. Initialization Card

Columns 15-21 - Blank

Column 22 - An I1 field containing a clue for the printing of a comprehensive table of member output data and a table of cap forces. Enter "1" if these tables are to be printed together with the other output data. Enter "2" if they are to be printed offline, using a tape generated for that purpose, and introduce appropriate instructions in the JCL deck. Leave blank (or enter "0") if only the condensed table of member output data is to be printed.

Column 23 - An I1 field containing a clue providing for the printing, in each cycle in the deflection-constraint mode, of a table listing derivatives of constrained deflection with respect to submember weights, submember gages before and after deflection-constraint resizing, and other related data. Enter "1" if the option to print out this table is exercised.

Column 24 - An I1 field containing a clue to require the application of "cutoffs" in the stress-constraint resizing of composite members. Enter "1" if the option to apply cutoffs is being exercised.

Column 25 - Blank

Columns 26-30 - Five fields (each I1) containing clues providing for various types of output on an optional basis, much of it useful primarily for debugging purposes. A value of "1", entered for any one of these clues, indicates that the specified output is requested; otherwise, the field should be left blank. The user is cautioned that the exercise of some of these options prints out very large amounts of data, much of it meaningful only to someone familiar with the program details. With the setting of each clue, the following output is obtained:

Column 26 - Intermediate output, including submember stiffness matrices, in each cycle in the stress-constraint mode. (A large amount of data is printed out.)

Column 27 - Intermediate data in the determination of the derivatives of constrained deflection with respect to submember weights in each cycle in the deflection-constraint mode. (A large amount of data is printed out.)

Column 28 - Submember corner forces (and moments) in each iteration cycle in both the stress-constraint and deflection-constraint modes. These forces (and

moments) are listed sequentially, starting with the first submember, and, for each submember, consist of the quantities shown in Appendix B, in the order shown there.

Column 29 - Nodal deflections following every structural analysis, in both the stress-constraint and deflection-constraint modes. These deflections are identified by degree-of-freedom number rather than node and component. In large problems, this option causes the printing out of a large volume of data.

Column 30 - Tape and file number locations for the important matrices, including those containing nodal deflections, submember corner forces, member data, etc., and also the execution times for various portions of the computations.

Columns 31-69 - Any alpha-numeric title for the problem, which will then be printed at the beginning of the output.

Column 70 - An I1 field containing a clue to print element edge shear flows obtained by differencing element nodal forces. Enter "1" if operative.

Column 71 - An I1 field containing a clue to calculate and print element warp loads (or kick forces) for elements of types 6 and 8. (This information is of very little use, and its calculation causes a significant increase in computational cost.) Enter "1" if operative.

Column 72 - An I1 field containing a clue to print noncomposite element stresses and stress resultants and composite element stress resultants in the property axis system. If this option is not exercised, the corresponding stresses and stress resultants will be printed in the local element axis system for each element. Enter "1" if operative.

Column 73-80 - Title of the member pseudo-matrix following redesign. This may be any alpha-numeric name. It is useful when final member data is to be used in another program.

Following the initialization card, there are three cards containing modification factors for allowable stresses. This input permits the user to modify the allowable stresses for isotropic and orthotropic materials, from the values supplied in the materials table or on member data cards, by a multiplication by the factors supplied, and to do so differently in different loading conditions. It is useful, for example, when the structural temperature is different for different loading conditions, but it must still be

uniform throughout the structure. There is one card each for tension, compression, and shear allowable stresses. When the material is orthotropic, the same factors are used for allowable normal stresses in both the x and y property axis directions. As shown in Figure 14, a 20F4.0 format is used on each card, to accommodate factors for up to twenty loading conditions. The values used should be less than 10.0. If the allowable stresses are not to be modified, three blank cards should be inserted in this location.

The cards that follow contain the member numbers of those members that are not to be candidates for resizing to satisfy the deflection constraint, if such a constraint is applied. As shown in Figure 14, these numbers are entered on the card (or as many cards as are needed) in a 16I5 format. The first field on any card containing this data must be filled, and no blank fields should be left between occupied fields on each card, but the fields need not be filled to the right end of any card. If any of the noncandidate members occur in groups of consecutively numbered members, only the first and last member numbers in any such group need be entered, with the last member number given a minus sign (having the meaning "through") and placed in the field immediately following the first member number. These data cards are followed by a single blank card. If no noncandidate members for deflection-constraint resizing are specified, only the single blank card is used.

4.2.3 Nodal Geometry and Boundary Conditions

The nodal geometry and boundary condition data are entered as shown in Figure 15. The node number (I4 format) is entered in columns 1-4, followed by three E13 fields for the x, y and z global coordinates. The boundary conditions are for the six degrees of freedom, Δ_x , Δ_y , Δ_z , θ_x , θ_y , θ_z , entered in columns 54 through 59, using the following clues:

- 0 (or blank) - zero displacement component. This clue causes the row and column for the particular displacement component to be removed from the structural matrices that are created. Alternatively, a "2" may be used in place of "0".
- 1 A "1" in any one of the six boundary condition columns indicates a "free" (not specified) degree of freedom.

The remaining columns on the card are not used.

[illegible]

Figure 14. Format for Allowable Stress Modification Factors and Noncandidate Members for Deflection-Constraint Resizing

The use of "2" rather than "0" as the clue in the case of zero displacement components is advantageous in certain situations. For example, when rotational degrees of freedom are not explicitly considered, the corresponding clues can be entered as zeroes or the appropriate spaces left blank. Where translational displacements are zero, the clue can be entered as a "2", as the program provides for counting the number of 1's and 2's entered, thus providing a check on the correctness of the boundary condition input.

The geometry and boundary condition data should be entered with the nodes in ascending numerical order and with no numbers missing. Nodes not connected to any element and with boundary conditions of 0, referred to as slack nodes, may be interspersed in the data, with the program effectively ignoring them. Should the user wish to modify the original idealization of a large problem, these additional nodes may be used and will be reasonably well-positioned numerically.

Geometry and boundary conditions are entered into the system in any one of the following ways:

- (a) The data, in the format indicated by Figure 15, is preceded by a LABEL (1) card, where:

NAMEA = name for the geometry pseudo matrix

NAMEB = name for the boundary condition pseudo matrix

- (b) The data, in the format indicated by Figure 15, is preceded by a LABEL (2) card, where:

NAMEA = name for the geometry pseudo matrix

Any data in the boundary condition fields will be ignored.

- (c) The data, in the format indicated by Figure 15, is preceded by a LABEL (3) card, where:

NAMEB = name for the boundary condition pseudo matrix

Any data in the geometry fields will be ignored

- (d) The boundary conditions are specified using a special condensed format, shown in Figure 16, where the "typical" nodal degrees of freedom are indicated and all exceptions are specified. This format is very useful

when the boundary conditions form a pattern that is very repetitive. The data is preceded by a LABEL (7), NAMEA card, where NAMEA is the name of the boundary-condition pseudo matrix. The first card indicates the standard degrees of freedom (columns 1 through 6 contain 0 (or 2) or 1 corresponding to the six degrees of freedom, Δ_x , Δ_y , Δ_z , Θ_x , Θ_y , and Θ_z). Columns 7 through 10 contain the total number of nodes in the structure. The remaining data cards indicate degrees of freedom that are exceptions to the standard. Twelve fields of I5 format may be used to record this information, with a minus sign indicating "through". For example, in Figure 16, a structure containing 324 nodes has standard degree of freedom 1, 1, 1, 0, 0, 0. The exceptions to this are indicated in the cards that follow; thus nodes 5, 8, 30 through 36, 80 etc., have degrees of freedom 1, 1, 0, 0, 0, 0. Note that the nodes do not have to be in sequence. However, a blank within the 12I5 data fields is not permitted; it will cause the remaining fields to be ignored. When using this format, all the nodes must appear in the geometry data in consecutive order, with no node numbers skipped.

4.2.4 Applied Loads

For purposes of inputting applied external loads, it is most desirable to work with physical designations, such as node number and component, rather than the row number of the load matrix. This is especially true when the structure has mixed nodal degrees of freedom and it becomes cumbersome to keep a count on the line-up of the degrees of freedom. The LABEL (6) card provides for inputting the load matrix, using physical designations rather than row numbers. The actual data is entered as shown in Figure 17. The following rules apply in using this data form:

- (a) On the LABEL (6) card, LABEL (6) is followed by the eight-character name of the pseudo matrix, which is followed immediately by the number of load conditions (columns) enclosed in parentheses. Using this type of data input (pseudo matrix), the program generates the actual load matrix.
- (b) One, two, or three fields may be used for the load data, starting at the left side of the form and working toward the right side (that is, by rows rather than columns).

- (c) The entire block of data must be entered in ascending order of the node numbers.
- (d) Within a given node, the components must be in the order and with the designations FX, FY, FZ, MX, MY, MZ. Alternatively, these designations may be entered as X, Y, Z, MX, MY, MZ, or simply 1, 2, 3, 4, 5, 6.
- (e) Within a given node and component, values for all loading conditions are entered together, with the condition numbers (column numbers) in ascending order.
- (f) Only non-zero values of the loads need be entered.

Figure 17 illustrates how the data should be entered. Note the "8" within the parentheses immediately following the matrix name on the LABEL(6) card. This indicates that the matrix contains eight conditions (columns). Note also that one, two, or three load values may be placed on a single card, and that the cards are in ascending order of node number, component, and condition number.

4.2.5 Material Properties

The program incorporates the properties of a small number of commonly-used structural metals. These are aluminum alloy, stainless steel, and titanium alloy 6-4, with the properties listed in Table 3, and with material code numbers as shown. The standard minimum gage for members of these materials is 0.01.

In addition to, or in place of, the standard materials, the user may introduce properties for materials in any or all of three classes of materials: isotropic, orthotropic, and composite. Properties for a maximum of 20 materials, including the standard materials (if they are not replaced), may be introduced.

If only the standard materials are to be used, the material properties deck may be completed excluded from the input data. If material properties are being introduced, the required data deck is preceded by a LABEL(4) card and followed by a blank card. This deck must precede the member data deck. The content and format of the data cards is shown in Figure 18 for the three classes of materials.

The material code (columns 7 and 8) is any integer from 1 to 20, identifying the material, and must appear on every card. The data class (column 15) identifies the material class. It is 1 for isotropic materials, 2 for orthotropic materials, and 6, 7 and 8 for composite materials. The data subclass (column 16) identifies the type of data, within each class, contained on the card.

TABLE 3. PROPERTIES OF STANDARD MATERIALS

Material Code	Material	Density lbs./in. ³	Elastic Modulus	Poisson's Ratio	Allowable Stresses		
					Tension	Comp.	Shear
1	Aluminum	0.100	1.05×10^7	0.3	67000	57000	39000
2	Steel	0.285	2.95×10^7	0.3	220000	213000	129000
3	Titanium	0.160	1.60×10^7	0.3	130000	127000	76000

Two cards (subclass 1 and 2) are used for isotropic materials, as shown in Figure 18a. These cards contain information defined as follows:

- ρ - material density
- t_{\min} - minimum gage
- t_{\max} - maximum gage
- R - Young's modulus
- γ - Poisson's ratio
- F_t - allowable tensile stress
- F_c - allowable compressive stress
- F_s - allowable shear stress

If the field for t_{\max} is left blank, no maximum gage limitation will be applied. It should be noted that the allowable stresses are all entered as positive quantities. All quantities entered are in F8.0 format. Columns 17 to 32 of the first card may be used for any alpha-numeric name identifying the material. If it is desired to apply cutoffs in resizing, as shown in Figure 2, the material can be introduced as an orthotropic material, with allowable stresses in the y-direction differing very slightly in value from those in the x-direction.

Three cards (subclasses 1, 2 and 3) are used for orthotropic materials, as shown in Figure 18a. These cards contain information defined as follows:

- ρ - material density
- t_{\min} - minimum gage

t_{\max} - maximum gage

E_{11} , E_{22} , G_{12} , ν_{12} - orthotropic elastic properties, as defined in Appendix VI, referred to property axes. (Subscripts 1 and 2 denote x and y property axes.)

F_{xt} , F_{xc} , F_{yt} , F_{yc} , F_s - allowable stresses referred to property axes. (Subscripts x and y denote property axes; subscripts t, c, and s denote tension, compression, and shear, respectively).

The comments following the definition of quantities on the isotropic material cards apply here as well.

As discussed in Subsection 2.1, a composite material is defined as being an entire laminate, with the numbers of laminae in the individual layers being the design variables. Accordingly, the definition of a composite material's properties requires the definition of the properties of the individual layers. Four cards are used to define the properties of each layer, as shown in Figure 18b. The properties of Layer No. 1 are entered on cards of data class 6, subclasses 1, 2, 3, 4; Layer No. 2 on data class 6, subclasses 5, 6, 7, 8; Layer No. 3 on data class 7, subclasses 1, 2, 3, 4; Layer No. 4 on data class 7, subclasses 5, 6, 7, 8; Layer No. 5 on data class 8, subclasses 1, 2, 3, 4; Layer No. 6 on data class 8, subclasses 5, 6, 7, 8. There can be any number of layers up to a maximum of 6. The cards should always be entered in the order shown in Figure 18b. If there are fewer than 6 layers, the first card should always be the class 6, subclass 1 card, and the remaining cards should be in the same order as in Figure 18b, without any omissions. The number of cards used should equal the number of layers in the laminate.

The quantities entered for each layer are defined as follows:

- l - initial number of laminae
- l_{\min} - minimum number of laminae
- l_{\max} - maximum number of laminae
- ϕ - angle between the property x-axis and the fiber direction in degrees. (Refer to discussion below.)
- B. L. Clue - balanced layer clue. (Refer to discussion below.)

MEMBER NUMBER	TYPE	MATERIAL	GRAPH CODE	CONSTR. CODE	DATA CLASS	NODES					FACTORS										12																	
						1	2	3	4	5	1	2	3	4	5	6	7	8	9	10		11	12	13	14	15	16											
01	03	05	07	08	15	17	19	21	23	25	27	29	31	33	35	37	39	41	43	45	47	49	51	53	55	57	59	61	63	65	67	69	71	73	75	77	79	81
02	04	06	09	10	16	18	20	22	24	26	28	30	32	34	36	38	40	42	44	46	48	50	52	54	56	58	60	62	64	66	68	70	72	74	76	78	80	
					61	Identification																																
					62																																	
					63																																	
					64																																	
					65																																	
					66																																	
					67																																	
					68																																	
					71																																	
					72																																	
					73																																	
					74																																	
					75																																	
					76																																	
					77																																	
					78																																	
					81																																	
					82																																	
					83																																	
					84																																	
					85																																	
					86																																	
					87																																	
					88																																	
01	03	05	07	08	15	17	19	21	23	25	27	29	31	33	35	37	39	41	43	45	47	49	51	53	55	57	59	61	63	65	67	69	71	73	75	77	79	81
02	04	06	09	10	16	18	20	22	24	26	28	30	32	34	36	38	40	42	44	46	48	50	52	54	56	58	60	62	64	66	68	70	72	74	76	78	80	

(b) Composite Materials

Figure 18. (cont'd) Format for Material Properties

- t - thickness of an individual lamina
- ρ - layer material density
- G_z - allowable stress in the microbuckling failure mode
(If this field is left blank, microbuckling will be ignored.)
- $E_{11}, E_{22}, G_{12}, \nu_{12}$ - orthotropic elastic properties of layer material
(including matrix), as defined in Appendix VI, referred to
fiber axes (Subscripts 1 and 2 denote coordinates along and
transverse to fiber direction, respectively)
- F_{x_t}, F_{x_c} - allowable stress in tension and compression, respectively,
along fibers (both positive quantities)

If the field for ℓ_{\max} is left blank, no upper limit will be placed on the number of laminae. The angle ϕ is defined as positive in the same sense as the element β - angle (angle between the local element x-axis and the property x-axis), that is, positive away from the element, as shown in Figure 1. Consequently, if this definition is to yield consistent results for all elements of a given composite material, the local element axis system, for all of these elements, must be consistently defined. This requires that there be consistency in the order in which each element's nodes are listed; for example, the i-j direction should either be clockwise, or counterclockwise, around all of these elements. The balanced-layer clue is a single-digit integer which should be given the same value for any two layers that are to be balanced. If more than two pairs of layers are to be balanced, a different value should be used for each pair. The field for the balanced-layer clue may be left blank for any layer that is not to be balanced. An F8.0 format is used for all quantities defined above, with the exception of $\ell, \ell_{\min}, \ell_{\max}$ and the balanced layer clue, which are integers.

It should be noted that the material properties input can be used to simplify and reduce the member data input. For example, if the same minimum or maximum gages are to be applied to all members of a given group, a specific material, with these minimum or maximum gages, can be defined for these members. It is then unnecessary to define these minimum or maximum gages in the member data. The same applies if gages are to be fixed at the same value for all members of a given group. The minimum and maximum gages are then both set equal to this fixed value in the material properties data.

4.2.6 Member Data

The member data deck is preceded by a LABEL (5) card and ends with a blank card.

If the β -angle (angle between the local element x-axis and property x-axis), for each member of any group of members, is to be computed internally, using a reference direction to determine the orientation of the property axes, as discussed in Subsection 2.1, the necessary data cards are introduced immediately following the LABEL (5) card. Reference directions may be defined for a number of zones (not to exceed 50). There is one data card for each zone, with format as shown in Figure 19. Each card contains the zone number and the x, y, and z coordinates, in the global axis system, of two points, A and B, defining the reference direction A to B. This reference direction, when projected into the plane of an element in the corresponding zone, defines the direction of the property x-axis.

The cards defining the reference directions should be followed by a blank card. If no reference directions are defined, a single blank card should be placed immediately following the LABEL (5) card.

A standard format is used for all member data cards. It has been designed to accommodate not only the finite elements used in ASOP-3, but also elements that may be added to the program in the future. This format is shown in Figure 20.

As the data for any one member may require more than one card, the various cards that may be used are distinguished from one another by the data class and subclass, entered in columns 15 and 16, respectively. There are eight data classes, identifying five basic categories of data, as follows:

- | | |
|----------|--|
| 1 | Topology and geometric properties |
| 2 | Elastic properties |
| 3 & 4 | Categories reserved for possible future use |
| 5 | Allowable stresses and minimum and/or maximum gages for noncomposite materials |
| 6, 7 & 8 | Composite material properties |

The subclasses further identify data within these categories.

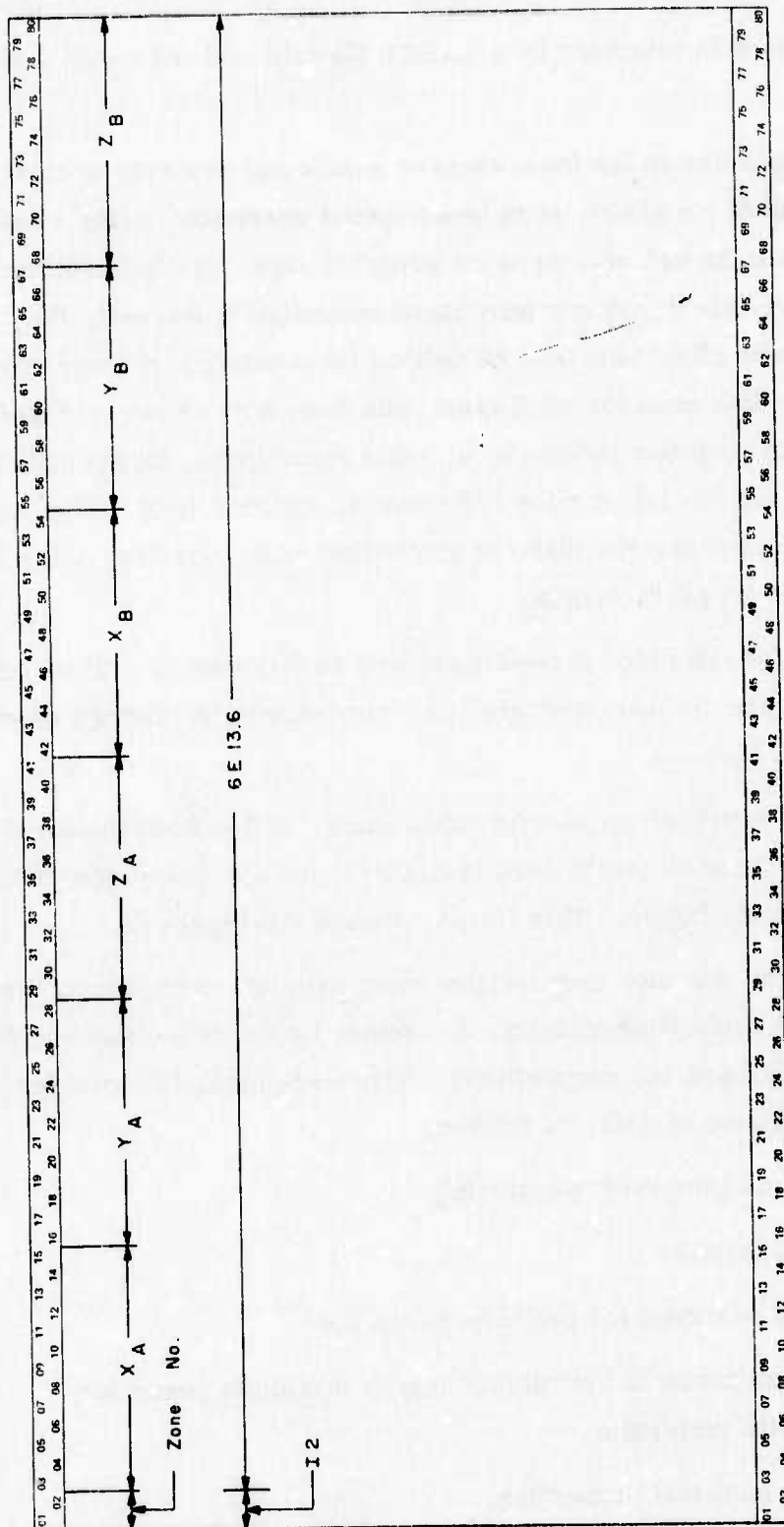


Figure 19. Format for Coordinates of Points Defining Reference Directions for Property Axes

The specific data entered on the cards of data classes 1 and 2 varies with type of element. It is shown in Appendix B for the various types of element used in ASOP-3.

A data class 1, subclass 1, card is required for every member and should always be the first card for each member. Frequently it is the only card required. It always contains the member number, member type, material code, data class and subclass, node numbers for nodes connected by the member, member initial gage (Factor 1), and such additional data as is needed for the type of member under consideration.

In the case of beam elements (types 2 and 11), care should be exercised in defining the angle β , which establishes the orientation of the y and z axes, as bending moments about the y-axis are disregarded in resizing. The xy plane should be the plane in which the element is primarily loaded in bending.

In the case of membrane elements (types 4, 5, and 8), for which the β -angle (angle between the local element x-axis and property x-axis) is to be computed internally, as discussed above, the zone number is entered in the Factor 5 field. It will identify the reference direction that is to be used. A value for the β -angle computed in this way will override any value entered elsewhere on the class 1, subclass 1, card. Accordingly, care should be taken not to enter a zone number unless no value for the β -angle is entered on the class 1, subclass 1, card, or any value entered is to be overridden by the internally-computed value. In the case of composite membrane elements, care should be taken in ordering the nodes, as discussed in Subsection 4.2.5.

A construction code (columns 13 and 14) is entered only if a stability table is to be used, as discussed in Subsection 2.3.4, and it then consists of the stability table number. If more than four nodes are used in defining the member, the additional node numbers are entered on a data class 1, subclass 2, card.

Cards of data class 2 are not needed if the elastic properties are those for the material specified by the material code. However, class 2 cards can be used to override those properties in the case of noncomposite materials or materials that are not explicitly treated as composites. The quantities to be entered are defined in Appendix B for the various element types. In the case of membrane elements (types

4, 5 and 8), they are the coefficients in the stress-strain relations, A_{11} , A_{22} , etc., referred to property axes. Normally, the elastic properties will be either isotropic or orthotropic with respect to the property axes, in which case only four of these coefficients need be defined and only a subclass 1 card need be used. However, provision is made for more generally anisotropic materials or materials that are orthotropic with respect to axes other than the property axes as defined. In that case, six coefficients must be defined, and a subclass 2 card is required in addition to the subclass 1 card. It should be noted that this greater generality is useful only for analysis purposes, as the resizing capability in ASOP-3 is applicable only to materials that are isotropic or are orthotropic with respect to the property axes.

Data class 5 cards are used to override allowable-stress values and minimum or maximum gages, as defined in the material properties data, for isotropic or orthotropic materials. The allowable stresses are entered on the subclass 1 card, and the minimum or maximum gages on the subclass 2 card, as shown in Figure 21, Sheet 1, for the two classes of materials. The definitions of the quantities entered are the same as in the material properties input (Subsection 4.2.5).

Cards of data classes 6, 7, and 8 are used for composite members, when it is desired to override data entered through the material properties input. The data class and subclass numbering system, and the quantities entered, correspond to those in the material properties input (Subsection 4.2.5), as shown in Figure 21b.

It should be noted that, when cards of data classes 5, 6, 7, and 8 are used, only those cards that actually contain data differing in value from the corresponding data in the material properties input need be included. However, whenever such a card is used, the data on it should be complete; otherwise, omitted quantities will be given a zero value. It should be noted also that all member data cards must include the member number (columns 1-4). All entries in the Factors 1-5 fields of the member data cards that are physical quantities should be in F8.0 format. All other defined quantities (including numbers of laminae and balanced layer clue), in the member data cards, should be in integer format.

The member data cards can be used to fix member gages. This is done by setting both the minimum and maximum gages equal to the desired value. If a substantial number of members are to be fixed at the same gage, it is more expeditious to do this through the material properties input, as discussed in Subsection 4.2.5.

MEMBER NUMBER	TYPE	MATERIAL CODE	GRAPH CODE	CONST. CODE	DATA CLASS	NODES				FACTORS										I.D.																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																						
						1		2		3		4		1		2		3		4		5		73	74	75	76	77	78	79	80																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																											
						17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	32	33	34									35	36	37	38	39	40	41	42	43	44	45	46	47	48	49	50	51	52	53	54	55	56	57	58	59	60	61	62	63	64	65	66	67	68	69	70	71	72																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																					
01	03	06	09	11	13	15	16	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48	47	46	45	44	43	42	41	40	39	38	37	36	35	34	33	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16	15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00	99	98	97	96	95	94	93	92	91	90	89	88	87	86	85	84	83	82	81	80	79	78	77	76	75	74	73	72	71	70	69	68	67	66	65	64	63	62	61	60	59	58	57	56	55	54	53	52	51	50	49	48

The only restriction on the order in which members are arranged in the member data deck is that discussed in Subsection 4.1. It is that any two members having one or more common nodes should not be very widely separated in the member sequence.

4.2.7 Deflection-Constraint Data

The deflection-constraint data is entered as shown in Figure 22. The first card in the deck is a LABEL (8), NAMEA card, as shown, and the last card should be a blank card.

The first card following the LABEL (8) card contains four quantities, defined as follows:

KLUGD is a clue specifying the type of constraint

KLUGD = 0 if $\delta = \delta_{\text{constraint}}$ (equality constraint)

KLUGD = 1 if $\delta \leq \delta_{\text{constraint}}$ (inequality constraint)

KLUGD = -1 if $\delta \geq \delta_{\text{constraint}}$ (inequality constraint)

where δ is the subject deflection and $\delta_{\text{constraint}}$ is the constraint value

CYCLES is the maximum number of iteration cycles in the deflection-constraint mode

ΔWT and $\Delta \delta$ are parameters in the convergence criterion for iteration in the deflection-constraint mode, defined as follows: The process will be considered to have converged if, in two successive cycles, the subject deflection is equal to the constraint value within a tolerance $\Delta \delta$ ($\delta_{\text{constraint}} - \Delta \delta \leq \delta \leq \delta_{\text{constraint}} + \Delta \delta$), and the total weight of the structure has not changed by more than ΔWT . If the fields for these parameters are left blank, the iteration process will proceed until the maximum number of cycles specified have been performed.

The next card contains four quantities, defined as follows:

NSTEPS is the number of steps into which the change from the initial value of the subject deflection to the vicinity of the constraint value is to be divided. If the field for NSTEPS is left blank, the program sets it equal to 1.

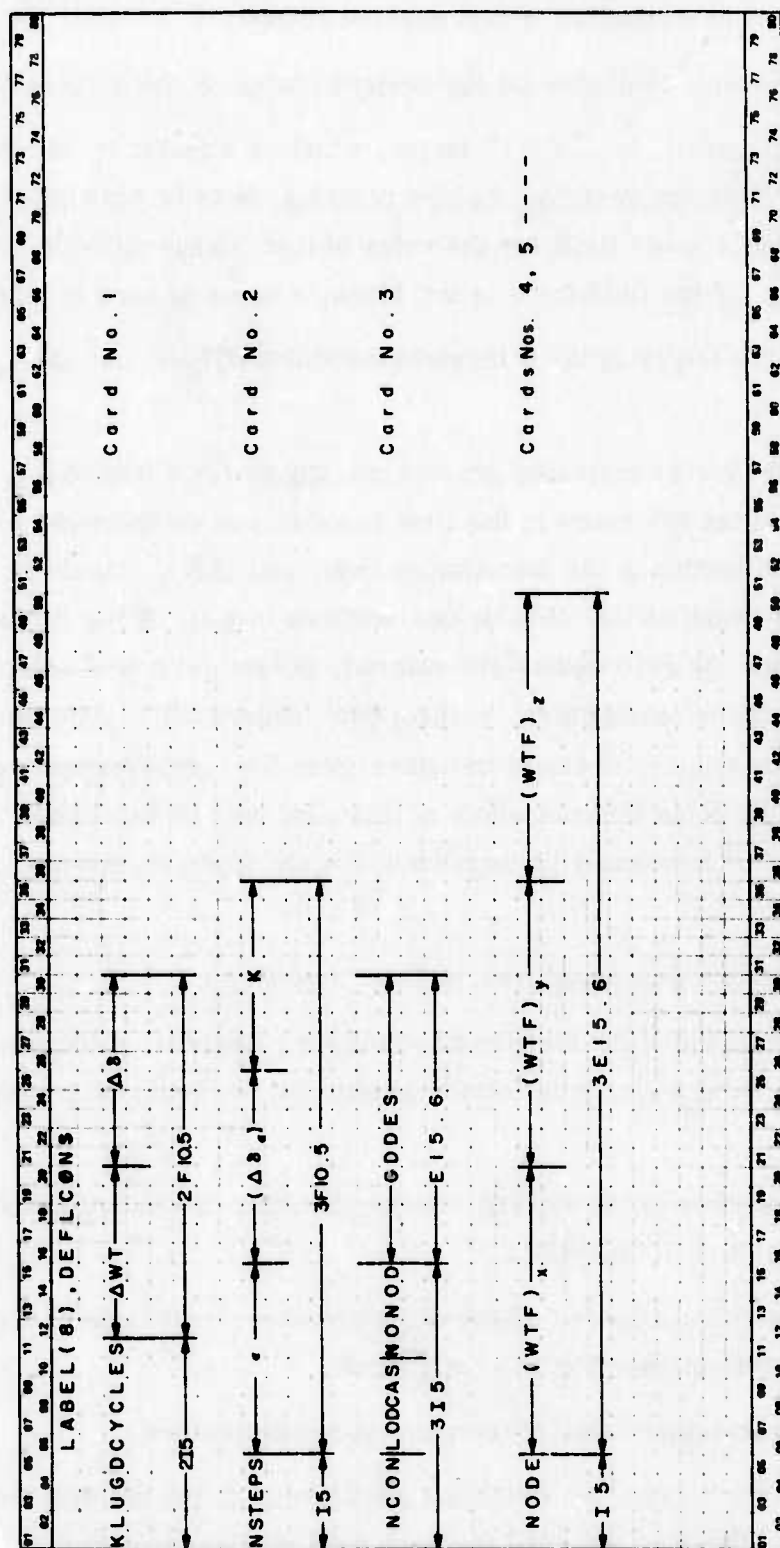


Figure 22. Format for Deflection-Constraint Data

- ϵ is the tolerance on the desired change in the subject deflection, expressed as a fraction of that desired change
- $(\Delta\delta)_\epsilon$ is the absolute tolerance on the desired change in the subject deflection
- k is the so-called "MAX CUT" factor, which is a factor by which the value of a design variable, before resizing, is to be multiplied to determine a lower limit for the value of that design variable after resizing. If the field for k is left blank, a value of zero is taken.

It should be noted that the larger of the tolerances established by ϵ and $(\Delta\delta)_\epsilon$ is used in the program.

A rough rule of thumb for selecting appropriate values for ϵ and $(\Delta\delta)_\epsilon$ is that ϵ should yield a value of the tolerance in the first cycle that is on the same order as $\Delta\delta$ (the tolerance on deflection in the iteration cycles), and $(\Delta\delta)_\epsilon$ should be considerably smaller than $\Delta\delta$ (such as one-fifth or one-tenth as large). If the fields for ϵ and $(\Delta\delta)_\epsilon$ are left blank, or zero values are entered, the program will select values of 0.01 and $0.1\Delta\delta$ for these parameters, respectively, unless $\Delta\delta$ is also zero, in which case the maximum number of trials in determining the target derivative will be set at 20. It should be noted that the whole of this card may be left blank, in which case default values will automatically be introduced for the four parameters, as discussed above.

The next card contains four quantities, defined as follows:

- NCON is the number of the constraint condition. Because of the present limitation to a single deflection constraint, it should be set equal to "1"
- LODCAS is the number of the applied loading condition in which the deflection constraint is to be applied
- NONOD is the number of nodes whose displacements participate in the generalized deflection subject to constraint
- GDDES is the constraint value of the generalized deflection

The remaining cards contain the weighting coefficients in the definition of the generalized deflection. There is one card for each node participating in that definition. Each card contains the node number and the weighting coefficients, $(WTF)_x$, $(WTF)_y$,

$(WTF)_z$, for the x, y, and z components of the nodal displacement in global coordinates, as shown in Figure 22. Blank fields may be left where the coefficients have zero values. If the constraint is simply on the translational displacement of a single node in a single coordinate direction, the generalized constraint becomes the constraint value of that displacement, and the corresponding single weighting coefficient is given the value 1.0.

As an example of the definition of a generalized deflection, consider the angular deflection constraint applied at the inboard station, in the model of the "Intermediate-Complexity Wing", shown in Figure 6. The streamwise line defining this station passes through upper-cover node 41 on the leading edge of the model and between upper-cover nodes 29 and 39 on the trailing edge of the model.

The angular deflection of this line, in degrees, can be represented as:

$$\alpha = 57.3 (\delta_{L_z} - \delta_{T_z}) / \ell$$

where δ_{L_z} and δ_{T_z} are the z-components of the translational displacements of points on the line at the leading and trailing edges of the model, respectively, and ℓ is the distance between them. δ_{L_z} and δ_{T_z} are given by:

$$\begin{aligned}\delta_{L_z} &= \delta_{41_z} \\ \delta_{T_z} &= k_1 \delta_{29_z} + k_2 \delta_{39_z}\end{aligned}$$

where δ_{41_z} is the z-component of the translational displacement of node 41, etc., and k_1 and k_2 are factors used in determining δ_{T_z} by a linear interpolation between δ_{29_z} and δ_{39_z} .

The angular displacement can now be written as:

$$\alpha = - (57.3 k_1 / \ell) \delta_{29_z} - (57.3 k_2 / \ell) \delta_{39_z} + (57.3 / \ell) \delta_{41_z}$$

where the coefficients of the displacements are the required weighting coefficients, $- (57.3 k_1 / \ell)$ being the value of $(WTF)_z$ for node 29, and so on. In this case, only the z-components of displacement are involved, so that the fields for $(WTF)_x$ and $(WTF)_y$ can be left blank.

As another example, the camber at this station could have been constrained. The generalized deflection representing camber could then be expressed as the

difference between the z-component of displacement of a point at the middle of the streamwise line and the average of the z-components of displacement of points on that line at the leading and trailing edges.

4.2.8 Stability Tables

As discussed briefly in Subsection 2.3.4, the resizing of bars and shear or compression panels that are subject to buckling failure may be accomplished by the use of "stability tables" relating allowable stresses to internal loads. The input to a small subsidiary program that generates these tables is described in Reference 3. The input involved in introducing these tables into the ASOP-3 program is also described in Reference 3.

4.2.9 Flexible Supports

A description of the input necessary to introduce the data for flexible supports, as discussed in Subsection 2.5, is provided in Reference 3.

4.3 DESCRIPTION OF OUTPUT

4.3.1 General Information

The printed output from the ASOP-3 program consists of the following major blocks:

- (a) All input data, including program clues, printed either in card image format or rearranged to be more readable.
- (b) Geometry, boundary condition, and member data following processing necessary to put it into a form appropriate for introduction into the finite-element subroutines.
- (c) Data subject to revision in each iteration cycle in the stress-constraint mode.
- (d) Data subject to revision in each iteration cycle in the deflection-constraint mode, if deflection-constraint resizing is done.
- (e) Data associated with the final design, including nodal deflections, member gages, weights, stresses, strains, internal loads, critical stress ratios, and critical load conditions.

The printing of some blocks of output data is optional on the part of the user, as discussed in Subsection 4.2. Some of this optional output simply provides more detailed information than is otherwise obtained, while the remainder of it is useful primarily for debugging purposes.

4.3.2 Input Data

Following the ASOP-3 logo, which includes the release date of the program version being used, the data included on the initialization card is printed out. This consists first of the problem title, then the various clues that are set on the initialization card, and finally the data associated with the convergence criterion in the stress-constraint mode.

The next block of data is a list of the allowable stress modification factors. If no values for these factors have been entered, values of 1.0 are automatically inserted and printed out.

Following this, there is a list of members that are not to be candidates for deflection-constraint resizing.

The next block of data is a table of node coordinates in the global system and boundary condition clues. This data is in card image format.

Following this is a table of applied loads, again in essentially card image format, with some spreading out of data in rows. This table is followed by a table labeled SUMMARY OF APPLIED LOADS, which lists, for each loading condition, the summation of all the loads in each global coordinate direction (FX, FY, FZ), and the summation of moments about the global axes (MX, MY, MZ). This data can be useful in detecting errors in loads data. It can also be used in checking whether the structure stiffness matrix is ill-conditioned, as discussed in Subsection 4.1.

The next block of data is the material properties, listed separately for isotropic, orthotropic and composite materials. The table for isotropic materials includes the data for the three standard materials in the program, unless this data has been overridden by input data. The data for each material in the table for orthotropic materials is printed out on two lines. There is a separate table for each composite material, containing the properties of each layer in the laminate. If there are no orthotropic or composite materials defined, the corresponding tables do not appear.

If the β -angles (angle between the local-element x-axis and property x-axis) for any members are to be computed internally, as discussed in Subsections 2.1 and 4.2.6, the data necessary to define reference directions for property axes is printed out next. It is in the form of coordinates of two points, A and B, on the line A to B defining the reference direction. This is done for as many zones as are defined.

The member data, in card image format, is printed out next. Immediately following it, the "NUMBER OF CORNER FORCES IN THE TOTAL STRUCTURE" is given. That number is the total number of member forces for all members in the model. Member forces are defined for each type of element in Appendix B. In the case of some elements (e.g., types 5, 6, and 8), they include edge shear flows in addition to corner forces, and may include kick forces (or warp loads) in warped elements (types 6 and 8), depending upon whether the option to calculate these forces is exercised (Subsection 4.2.2).

If a deflection constraint is to be applied, the data for it follows next. Under the label "DEFLECTION CONSTRAINT RESIZING CRITERIA", the parameters in the convergence criterion are listed first, followed by the number of steps, or iteration cycles, to bring the subject deflection to the vicinity of the constraint value. Parameters used in establishing when to terminate the trials in the determination of the target derivative are listed next, followed by the "max cut" factor (Subsection 4.2.7). Under the label "GENERALIZED DEFLECTION CONSTRAINTS", the data that directly describes the constraint is listed. Immediately following the constraint value, a verbal message is printed out in parentheses. It indicates whether the constraint is an "EQUALITY CONSTRAINT", or, if it is an inequality constraint, whether the constraint value is "NOT TO BE EXCEEDED" or the subject deflection is "NOT TO BE LESS THAN" the constraint value.

After all input data has been read in, the message "MATRIX INPUT IS COMPLETE" appears.

The next data blocks printed out consist of input data that has undergone processing preparatory to the formation of element stiffness matrices and the stacking of the structure stiffness matrix. The first block includes a rewriting (in E-format) of the coordinates of the nodes in the global system. More importantly, under the heading "BOUNDARY CONDITIONS", and the subheading X, Y, Z, MX, MY, MZ, indicating coordinate directions for translation and rotation, respectively, it lists the number

assigned to each of the degrees of the freedom in the structure. It thus establishes the correspondence between degree-of-freedom number on the one hand and node and coordinate direction on the other. This information is needed by the user in reading nodal deflection data printed out optionally in each iteration cycle, as the deflections are listed by degree-of-freedom numbers rather than nodes and coordinates. It is needed, for the same reason, in reading other optional output data useful for debugging purposes. Zero and negative entries, under the heading BOUNDARY CONDITIONS, indicate coordinate directions in which nodes are fixed.

The next block of output information consists of a listing of all the element types included in the program's element library. It provides a key to the table that follows, as it lists the items of data printed in that table for each type of element. These items are identified by numbers in parentheses and corresponding descriptive names. For the user who is interested in becoming familiar with the program code, it is noted that these numbers correspond to columns in the member pseudo-matrix, the array in which member data is stored.

The table that follows next is a listing of the members, with most of the input data listed earlier in card-image format, but now in a different format. This data serves to check whether the card input was stored and retrieved properly. In addition, for noncomposite membrane elements (types 4, 5, and 8), it lists the stiffness coefficients A_{11} , etc., in the transformation of strains to stresses in the property axis system. It also indicates, for warped elements (types 6 and 8), the amount of warpage. This is measured as the mutual separation of the diagonal lines joining opposite corners. In the case of composite members, it would be cumbersome to include the much larger amount of detail necessary to describe these members. Consequently, under item (26), which is normally the member gage, the composite material number is given, and the user should refer to the material properties table for the necessary information. If any of the materials data has been overridden by member data cards, the first table of member data should be consulted for such information. The stiffness coefficients, A_{11} , etc., are not listed for composite members. At the end of this table, the total number of members in the model and the bandwidth of the structure stiffness matrix, following application of boundary conditions, are indicated.

If nodal deflections are not to be printed out in each iteration cycle, the next data appearing is the "FORCE DIRECTION TABLE"; otherwise, it appears after the nodal deflections for the initial design. This table indicates the direction of each of the summed internal forces (cap forces) acting at each node and assigns a number to it. Not all these numbers are listed in this table. The column on the right lists only the number of the first force indicated at each node. These numbers correspond to the numbers used later in the table of cap forces, as discussed in Subsection 4.3.5. It should be noted that some of the numbers assigned in the force direction table are associated with moments rather than forces, when beam elements are included in the model, as discussed in Subsection 4.3.5.

4.3.3 Cyclic Data in the Stress-Constraint Mode

When the option to print out nodal deflections in each iteration cycle is exercised (Subsection 4.2.2), the table listing these deflections appears ahead of any other data for the current cycle. The deflections are listed by degree-of-freedom numbers, with loading conditions arranged in columns.

Iteration cycles in the stress-constraint mode are numbered in the following manner: REDESIGN CYCLE NO. 0 refers to the initial design before any resizing is done, REDESIGN CYCLE NO. 1 refers to the first redesign, and so on, until all cycles in the stress-constraint mode have been performed. In each cycle, after the heading indicating the redesign cycle number, there is a listing of current member gages (layups in the case of composite members), which would be the initial gages in REDESIGN CYCLE NO. 0, the first resized gages in REDESIGN CYCLE NO. 1, and so on. If a deflection constraint is to be applied later, the next data listed consists of the constraint value of the deflection (DESIRED VALUE OF GENERALIZED DEFLECTION), the value of the subject deflection for the current design (CURRENT VALUE OF GENERALIZED DEFLECTION), and the departure of that current value from the constraint value (DEPARTURE OF GENERALIZED DEFLECTION FROM DESIRED VALUE). In addition, the ratio of the subject deflection to the constraint value is listed for both the design existing in the preceding cycle and the current design (only the current design in REDESIGN CYCLE NO. 0). Following this, there is a listing of the maximum value of the stress ratio for all members and all loading conditions. This value is listed for both the design existing in the preceding cycle and the current design, and, in the latter case, the member in which this maximum value

occurs is also identified. This is followed by the total weight of the structure in both the preceding and current cycles. In REDESIGN CYCLE NO. 0, there is no preceding cycle, so that only values for the current cycle are listed. After all the data for the current cycle has been listed, the redesign cycle number is repeated, as an aid in identifying the cycle when the data for each cycle extends over several pages.

At the end of the last iteration cycle in the stress-constraint mode, established as such either by satisfaction of the convergence criterion or completion of the specified maximum number of cycles, the message ITERATIONS IN STRESS CONSTRAINT MODE NOW COMPLETE is printed out. If a deflection constraint is to be applied, one of two messages is then printed out. If the deflection constraint is either 1) an equality constraint, or 2) an inequality constraint that is violated by the final design in the stress-constraint mode, the message is ENTERING DEFLECTION CONSTRAINT MODE. If the deflection constraint is an inequality constraint, that is satisfied by the final design in the stress-constraint mode, the message is DEFLECTION CONSTRAINT NOT VIOLATED - DEFLECTION CONSTRAINT MODE NOT ENTERED.

4.3.4 Cyclic Data in the Deflection-Constraint Mode

When the deflection-constraint mode is entered, the iteration cycle numbering is started again at REDESIGN CYCLE NO. 1. Within each cycle, the following blocks of data are printed out in the order shown:

- (a) A block of data listing the total weight of the design upon entry into the current cycle and the constraint value of the deflection, along with the current value of the subject deflection and the departure of the constraint value from it; current referring to values associated with the design existing upon entry into the current cycle. In the first cycle in the deflection-constraint mode, this design is the same as the design analyzed in the last cycle in the stress-constraint mode. In subsequent cycles, it is the design resulting from stress-resizing in the latter part of the preceding cycle.
- (b) A block of data that lists first the desired change in deflection in the current cycle and the bounds placed on it by the applicable tolerance. This desired change in deflection will differ from the discrepancy between the current value of the subject deflection and the constraint value only in the first $N-1$ cycles in the deflection-constraint mode, where N is the

number of steps to move from the value at entry into the deflection-constraint mode to the vicinity of the constraint value. Following this, there is a table listing data associated with each trial in the search for an appropriate value of the target derivative. This data includes the value of the target derivative, the change in the subject deflection associated with changes in the design variables in the current trial, and the corresponding change in the total structure weight. It will be noted that the last two items are termed "approximate". This is because they are based on values of the design variables before rounding, in the case of composite-element layers, and because the deflection change is based on the second order approximation given in Equation (2.9) of Subsection 2.4.2. The last value of the target derivative listed is the value that is used in deflection-constraint resizing. The last value of the approximate deflection change listed should be within the bounds specified in the upper part of this block of data.

- (c) If the option to print out a table of the derivative, $\left(\frac{\partial \delta}{\partial w_i}\right)_{\text{old}}$, etc., for each element, is exercised, that table is printed out next. It lists, for each member that is a candidate for resizing in the deflection-constraint mode, the derivative $\left(\frac{\partial \delta}{\partial w_i}\right)_{\text{old}}$, which is the derivative of the subject deflection with respect to member weight for the design existing at entry to the current cycle, the member gages prior to and following deflection-constraint resizing (OLD GAGE and NEW GAGE), the corresponding change in member weight (DELTA WT) and the contribution of that member's resizing to the change in the subject deflection (DELTA DISP), and, finally, a clue indicating whether the new gage was governed on the one hand (clue value = 1) by the deflection constraint, "max cut", or specified maximum gage, or, on the other hand (clue value = 0), by stress constraint or specified minimum gage. In the case of composite members, there is a separate line printed out for each layer, in the order in which the layers are numbered (although layer numbers do not appear in the table), since each layer is resized separately. The derivatives are then with respect to layer weight, and the gages are expressed as numbers of laminae (which means that rounding has been done). Weight and displacement changes are for layers after rounding. It should be noted that, in the first cycle in the

deflection-constraint mode, the old gages are those associated with the design analyzed in the last cycle in the stress-constraint mode. In subsequent cycles, however, they are the gages resulting from stress-constraint resizing in the latter part of the preceding cycle, and are not listed elsewhere.

- (d) If the option to print out nodal deflections in each iteration cycle is exercised, a table of such deflections, for the design resulting from deflection-constraint resizing in the current cycle, is printed out next.
- (e) The next block of data has the same form as that described in Subsection 4.3.3 for the stress-constraint mode. It lists data for a current design, that is the design resulting from deflection-constraint resizing in the current cycle. It should be noted that the member gages listed here will be the same as the "NEW GAGES" listed in the table described in item (3), except that they will also include the gages of those members that have been specified as being noncandidates for deflection-constraint resizing. It should be noted also that the quantities listed as applying to the "PRECEDING CYCLE" are those that were calculated at the same point in the preceding cycle, that is, immediately following deflection-constraint resizing. The one exception to this is in the first cycle in the deflection-constraint mode, where PRECEDING CYCLE refers to the design analyzed in the last cycle in the stress-constraint mode.
- (f) If the option to print out nodal deflections in each iteration cycle is exercised, a table of such deflections for the design resulting from stress-constraint resizing in the current cycle, is printed out next. It will be noted that a table of deflections is printed out twice in each cycle in the deflection-constraint mode: first, following the analysis of the design resulting from deflection-constraint resizing, and later, following the analysis of the design resulting from stress-constraint resizing.
- (g) In the final cycle in the deflection-constraint mode (established as final either by the convergence criterion or the specified maximum number of cycles, whichever governs), there is an additional block of data, of the form described in Subsection 4.3.3 for the stress-constraint mode. It is for the design that exists following stress-constraint resizing in that

final cycle. It will be noted that quantities listed as applying to the "PRECEDING CYCLE" are the same as the corresponding quantities listed in the similar data block for the design existing immediately prior to stress resizing in the final cycle in the deflection-constraint mode. In both blocks, they apply to the second-to-last cycle.

4.3.5 Final Data

In the process of determining member stresses for the final design, new member gages are determined on the basis of those stresses. The new design produced in that way is, however, not analyzed. While that design is, in effect, disregarded, it may be of some value to the user, particularly in assessing the extent to which convergence has been achieved in the iteration process. Accordingly, the member gages obtained are printed out in a table that follows the data for the final design in the last iteration cycle. A subheading of that table identifies the design as one for which no analysis has been done.

The next block of data printed out consists of the nodal deflections for the final design (the last design analyzed). It is in a format different from that of the table of nodal deflections printed out optionally in each iteration cycle, as discussed above. Here, it is arranged by nodes, and, within the print-out for each node, rows correspond to displacement components in the global system (possibly including angular displacement components) and numbered columns to loading conditions.

This is followed by blocks of data consisting of member output (gages, stresses, etc.) and internal or "cap" forces. As some of this data is very extensive, and can, for large problems, produce a very large volume of output, various output options are provided, as follows:

- (a) The member data is printed out in two different tables: one a "comprehensive table", providing information in considerable detail; the other a much more compact or "condensed table", providing more limited information. A table of cap forces and corresponding stresses is also printed out.
- (b) Only the "condensed table" of member data is printed out, but the comprehensive table and the table of cap forces are stored on tape, available for later printout. Special instructions are required in the JCL deck when this option is exercised.

- (c) Only the "condensed table" of member data is printed out, and the "comprehensive table" and the table of cap forces are not saved.

When option 1 is selected, the "comprehensive table" and the table of cap forces are printed out first, followed by the "condensed table". When the problem is of modest size, the "comprehensive table" is in one block, and is followed by the table of cap forces. In the case of large problems, however, the cap forces are formed in blocks, as explained in Subsection 4.1, and these blocks are printed out so that each follows a corresponding block of member data in the "comprehensive table". Thus, the blocks of cap forces are interspersed among blocks of member data in the "comprehensive table". This arrangement of the "comprehensive table" of member data and the table of cap forces is the same if they are stored on tape.

A description of these output tables is now provided. The "CONDENSED TABLE OF MEMBER OUTPUT DATA" is described first. It is arranged in columns and rows, with all the data for a noncomposite member being contained in a single row, and the data for each layer of a composite member being contained in a single row. The following items of data are listed by column:

- (a) Member Number
- (b) Member Type
- (c) Material Code
- (d) Node Numbers
- (e) Planform Area of Two-Dimensional Member or Length of Bar or Beam
- (f) Final Gage (Layer Number and Number of Laminae in the case of composite element layers)
- (g) Critical Loading Condition
- (h) Stress in Critical Loading Condition (σ_x , σ_y , τ_{xy} for a noncomposite membrane element, in property or local element axes, depending upon the clue in column 72 of the initialization card)
- (i) Stress Ratio in the Critical Loading Condition
- (j) Code for Governing Constraint (listed for each layer of a composite member)

- (k) Critical Load Condition for Microbuckling
- (l) Microbuckling Stress Ratio in Critical Load Condition.

In the case of composite members, the critical loading condition, stress, stress ratio and governing constraint, as listed for each layer, are values determined in the analysis of the composite with matrix stiffness neglected.

The "comprehensive table" of member data is given the heading "MEMBER OUTPUT DATA", which is immediately followed, on the right side of the sheet, by a number code indicating the type of constraint governing the design of each member (or each layer of a composite member). This is followed by column headings for loading condition, stress components, strain components, and a final column relating to shear panels (element type 6), bar members (element type 1), or beam members (elements type 6 and 11). This last column lists the shear flow for a shear panel and the axial load for a bar or beam member. The column headings are printed just once, at the top of the table, and the data that they identify is interspersed among other data defined directly where it is introduced.

Each member is identified by number, type, the nodes it connects, material code, area or length, gage (in the final design), and the angle β between the local element axes and property axes. In the case of noncomposite members, this is followed by identification of the critical loading condition for that member, the corresponding value of the stress ratio, and the code number identifying the constraint governing the design of that member. This is immediately followed, for such members, by the stress and strain components, referred either to property axes or to local element axes, depending upon whether the clue for property-axis printout has been inserted in column 72 of the initialization card. For beam elements, the stress printed out for each loading condition is the most critical of the four stresses calculated (the extreme-fiber stresses at each end). These stresses are for combined axial load and bending moment about the z-axis, and, as explained in Subsection 2.3.2, are the stresses used in resizing. In the case of shear webs, bar elements, and beam elements there is a final column listing the shear flow or axial load, as discussed above. In the case of beam elements, this data is followed by a table which lists, under the label "BEAM SHEARS AND MOMENTS", the beam shears in the y and z directions (VY and VZ), the torsional moment (MX), and the bending moments about the y and z axes at the two ends (MYI, MZI, MYJ, MZJ), for all loading conditions. In the case of non-

composite membrane elements, the stress and strain data is followed by a listing of the components of the stress resultant, with the axis system again depending upon the clue entered in column 72 of the initialization card.

The stress and strain data is in considerably expanded form for composite membrane elements. First, there is a listing of the components of stress and strain in each layer, referred to fiber axes (with the x-axis in the fiber direction), yielded by an analysis in which the stiffness properties of the matrix material are included. This is followed by a listing of the components of the stress resultant and strain for the whole laminate, referred either to property axes or local element axes, depending upon the clue in column 72 of the initialization card. Following this is a listing of the maximum principal compressive stress for all loading conditions, the corresponding stress ratio for microbuckling, and the loading condition that is critical for microbuckling. The user can ascertain whether microbuckling was critical for the member by looking at the critical constraint code number for layer number 1 in the following table. Finally, there is a table captioned, "MEMBER STRESS RESIZING DATA", which lists, for each layer, the number of laminae in the final design, the critical load condition, the critical stress when the laminate is analyzed with the stiffness of the matrix material neglected (this stress being necessarily in the fiber direction), the ratio of this stress to the allowable stress in the fiber direction, and the code number identifying the constraint governing the design of that layer.

The table of cap forces, which are the forces obtained by summing element forces at a particular node in the direction of straight lines joining that node to adjacent nodes, is arranged as follows:

- (a) The first column lists the identifying numbers assigned to the forces in the Force Direction Table, discussed earlier.
- (b) The second column identifies the node at which a force acts, and the third column identifies the direction of the force by identifying the adjacent element through which its line of action passes.
- (c) The fourth and fifth columns identify the loading conditions in which the maximum and minimum (or maximum negative) values of the force occur.

- (d) Succeeding columns list values of the forces, which are positive for tension and negative for compression. The information in parentheses following each force value identifies the loading condition number.
- (e) Following the listing of the values of each cap force, the direction of the force is established further by listing direction cosines (DX, DY, DZ) referred to global axes.

It should be noted that, when the model includes beam elements, some of the rows of the table of cap forces list bending moments rather than forces. This occurs at nodes at which beam elements are attached. For such a node, and for a direction in which a beam element lies, there will then be two rows in the table, the first being for the force in that direction and the second being for the bending moment about the z axis of the beam element.

When the option to calculate and print warp loads or kick forces is exercised (column 71 on the initialization card), the table containing this data is printed out next, preceded by a force direction table that serves a similar purpose to the corresponding table for cap forces. The arrangement of the warp load table is similar to the cap force table. It lists the summation of the warp loads, at any node, for all type-6 or type-8 elements adjacent to that node that have additional nodes m, n, o, and p defined (Appendices A. 6 and A. 7).

When the option to print shear flows acting on the edges of membrane elements is exercised (column 70 on the initialization card), the table containing this data is printed out next. These shear flows are based on differences between element nodal forces at the two ends of the edge and acting in the direction of the edge. The format resembles that for the tables of cap forces and warp loads, except that the data listed is for individual elements rather than being summed quantities.

Immediately following the "condensed table" of member output data, a list of those members that are overstressed (stress ratio exceeds 1.0) in at least one loading condition is printed. If the number of such members exceeds fifty, only the first fifty are printed out, followed by a message indicating that the number exceeds fifty.

The last table printed is a summary of the member gages and weights. If only stress-constraint resizing has been done, this table lists the gage (or layup, in the case of composite members) and the weight of each member in the final design. The total weight of the final design and the number of iteration cycles to reach it are indicated at the bottom of the table. If deflection-constraint resizing has also been done, this table lists the member gage and weight data for two designs: the design existing at the end of the stress-constraint mode, and the final design at the end of the deflection-constraint mode. In addition, it lists the change in weight of each member from the former to the latter design and the percentage distribution among members of the change in the total structure weight. In the case of composite members, this percentage distribution is broken down by layer. At the end of the table, there is an indication of the total weight of each of these two designs and the number of iteration cycles to reach it, as well as the change in total structure weight between them.

4.4 CAUTIONARY NOTES

In the preceding sections, the ASOP-3 program user has been cautioned, at various points, concerning model design, the use of program clues, and the preparation of input data to avoid pitfalls and ensure satisfactory performance of the program. These precautions are summarized below for the convenience of the user.

- (a) The use of triangular elements should be avoided wherever feasible, but, if required, they should be as nearly equilateral in shape as feasible. Quadrilateral elements should be as nearly rectangular as feasible, with an aspect ratio preferably not exceeding 2.
- (b) The JCL deck used should be consistent with the clue entered in column 22 of the initialization card for options associated with output data (Subsections 4.2.1 and 4.2.2).
- (c) The clues in columns 26, 27, and 30 should be used only by those familiar with the ASOP-3 program code. It is recommended also that the clues in columns 28 and 29 should not normally be used, because they produce a large volume of data that is usually of secondary value.
- (d) A clue must be entered in column 72 of the initialization card, if stresses and stress resultants are to be printed out in property axes, rather than local element axes.

- (e) The number of nodes in the structural model should not exceed 1000 and should be numbered consecutively, starting at 1.
- (f) To minimize the bandwidth of the structure stiffness matrix, and consequently to minimize computing time, nodes should be numbered to minimize the separation of any pair of connected nodes in the numerically ordered node list.
- (g) The number of loading conditions for analysis or stress-constraint resizing should not exceed 20. A deflection constraint should be applied in only one loading condition.
- (h) The number of members in the model should not exceed 3000. Furthermore, if a deflection constraint is being applied, the total number of submembers that are candidates for deflection-constraint resizing should not exceed 6000, where a submember is the same as a member in the case of noncomposite members and is a layer of a composite member. This latter requirement can be made less restrictive by the appropriate removal of members from candidacy for deflection-constraint resizing, as explained in Subsections 4.2.1 and 4.2.2.
- (i) The members cannot be ordered arbitrarily. Members that have common nodes should not be too greatly separated in the sequence. A rough rule-of-thumb is that such members should not be separated by more than 100 intervening members.
- (j) Care should be taken to insert blank cards wherever they are required in the input deck, as shown in Figure 12 and explained in Subsection 4.2. In particular, three blank cards should be inserted in place of those for "allowable stress modification factors", if allowable stresses are not to be modified. Immediately following the LABEL (4) card, a card or cards should be inserted, defining reference directions for property axes, followed by a blank card. If this data is not entered, only the single blank card that would normally follow it should be used.
- (k) The LABEL (6) card, preceding the applied load data, must have the number of loading conditions entered on it, as explained in Subsection 4.2.4, and this number must be the correct number.

- (l) In ordering nodes for composite members, there should be consistency among members in a given zone (all clockwise or all counterclockwise), as discussed in Subsection 4.2.5.
- (m) The use of the program on an IBM system may give erroneous results in some cases, because of insufficient precision in the calculations, as discussed in Subsection 4.1.

Appendix A

DESCRIPTION OF FINITE ELEMENTSA.1 INTRODUCTION

The finite elements in the ASOP-3 program are described in this section. Each element is shown in a sketch, and the assumptions used in formulating the element stiffness and stress matrices are given. For elements such as the bar and the beam elements, the discussion is kept brief, as details are available in many references. More lengthy descriptions are provided for those elements that are not completely described in the literature.

A.2 ELEMENT NO. 1 (Bar)

Figure A-1 shows a typical bar element which connects nodes i and j . The element carries only axial load, and $P_i = -P_j$, as a state of constant strain is assumed within the element. In addition, the cross-sectional area and material properties are constant along the length of the bar. A detailed derivation of the stiffness matrix can be found in Reference 6.

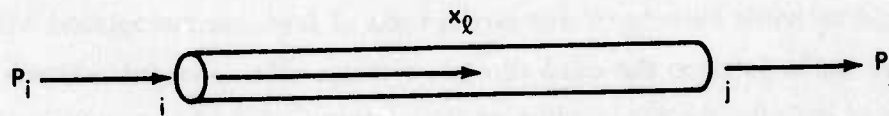


Figure A-1. Bar Element

A.3 ELEMENT NO. 2 (Beam)

The beam element has uniform properties along its length and is capable of resisting axial forces, bending moments about the two principal axes of the cross section, and twisting moments about the centroidal axis. In the case of a beam element with zero offset, as shown in Figure A-2, the position of the element and the orientation of its cross-sectional axes in space are specified by giving the coordinates of nodes i , j , and k and the angle β . Nodes i and j define the end points of the element, along its centroidal axis, and are also its points of attachment to the adjoining structure. Node k is an additional node which, together with nodes i and j , establishes a reference plane, while β is the angle between this reference plane and the plane containing the beam centroidal axis and the y_ℓ -axis; y_ℓ being a principal axis of the cross section. The sign convention for β is such that β is positive when a right-hand rotation about i - j (x_ℓ -axis) brings the former plane into coincidence with the latter plane.

As the bending moment about the y_ℓ -axis is disregarded in the resizing of this element, as discussed in Subsection 2.2.2, it is important to define the axes such that the x_ℓ y_ℓ plane is the plane in which the element is primarily loaded in bending.

When the beam element is offset, two additional nodes, ℓ and m , which now define the end points of the element along its centroidal axis, are introduced. The reference plane is now defined by k , ℓ , and m , and β is now the angle between that plane and the plane containing the centroidal axis and the y_ℓ axis, defined positive as before. Nodes i and j are now the points of attachment of the element to the adjoining structure.

It should be noted that there are no degrees of freedom associated with nodes k , ℓ , and m (or node k , in the case of zero offset). They are introduced purely for purposes of defining the kinematics of the element, and are treated in the same way as fixed nodes in the boundary condition input, even though they clearly do move when the structure is deformed. This movement is determined on the basis that ℓ is rigidly connected to i , and m to j .

The displacement functions selected for the bending deformations are cubic polynomials which are based on a beam theory neglecting shear deformation. Axial deformation and rotation about the centroidal axis are linear functions along the

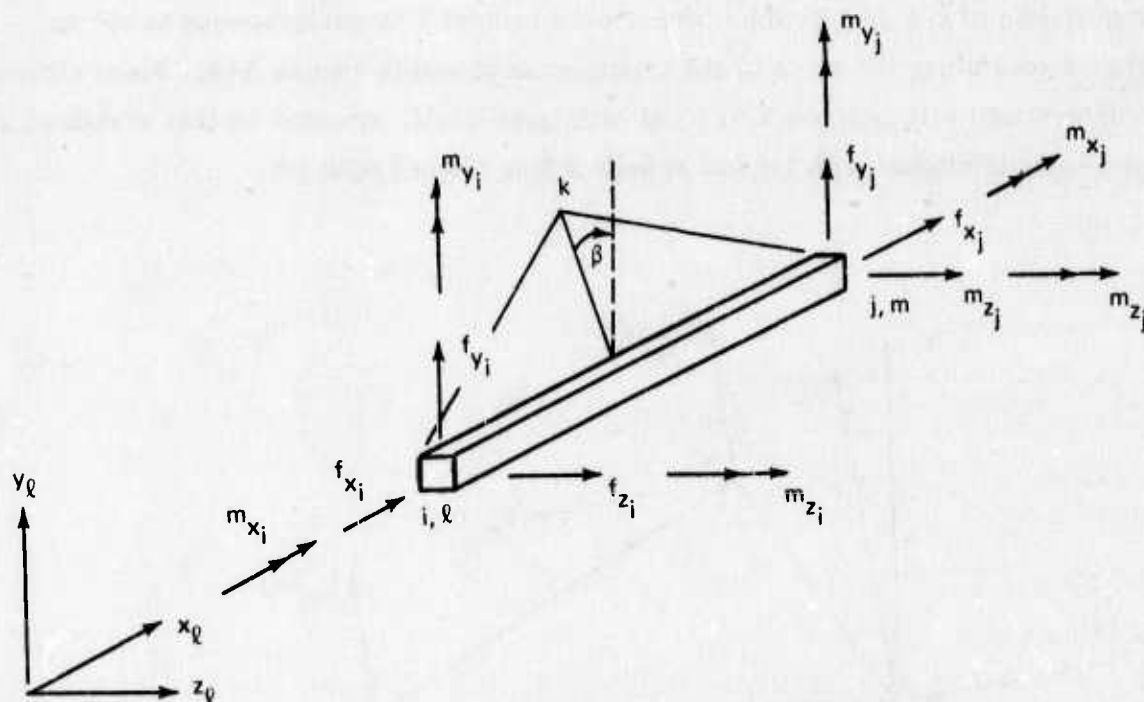


Figure A-2. Beam Element

length of the beam, giving rise to constant axial strain and uniform twist per unit length. These displacement assumptions are used in both References 5 and 6, which give a complete development of the stiffness and stress matrices for the beam element.

A.4 ELEMENT NO. 4 (Plane Stress Triangle)

The triangular membrane element, shown in Figure A-3, is based on the assumption of constant strain within the element, which results in compatible displacements along the boundaries of adjacent triangles. A general form of Hooke's law is employed, which allows for anisotropic, orthotropic, or isotropic material behavior. The material properties are defined with reference to the property axes (x_p, y_p) whose orientation, with respect to the element, is given by the angle β , which is defined positive as shown.

Derivations of the stiffness and stress matrices are given in Reference 5. The stress matrix is formed so that forces at the three nodes of the triangle are produced. For purposes of resizing by the nodal-stress method it is advantageous to obtain corner forces along the sides of the triangle, as shown in Figure A-4. Nodal stresses are determined with reference to local orthogonal axes, oriented so that at nodes i and j the x-axis is aligned with i-j and at node k it is aligned with j-k.

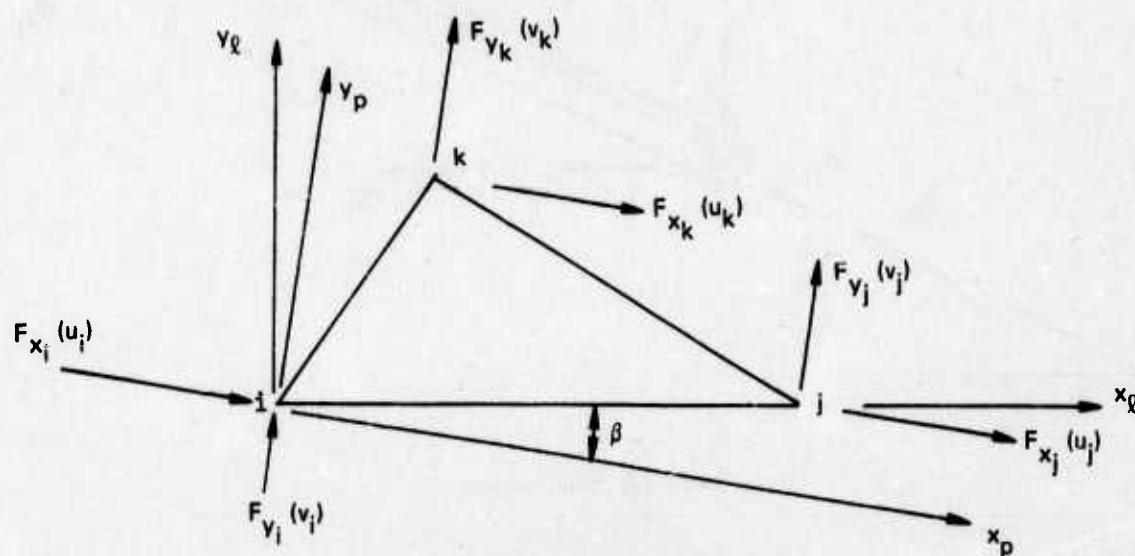


Figure A-3. Triangular Membrane Element

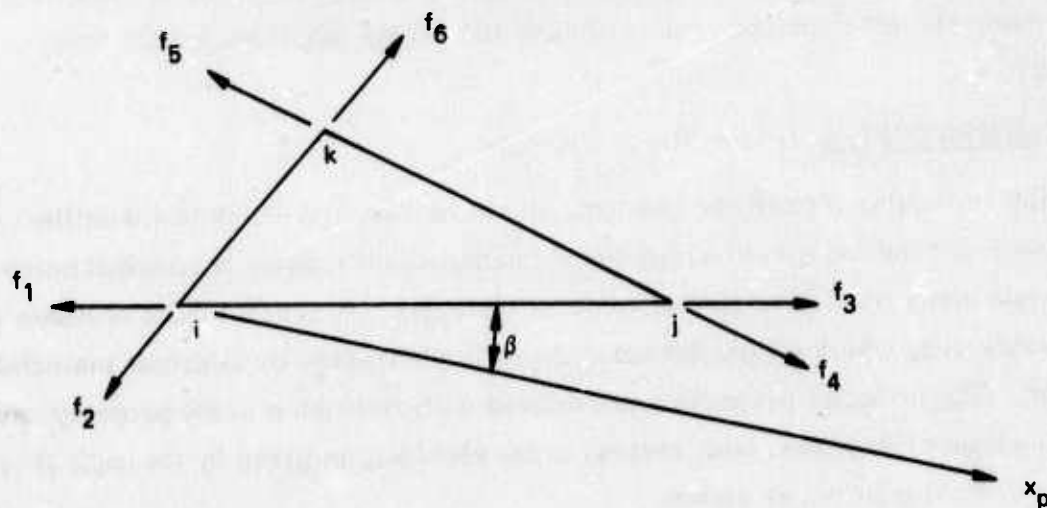


Figure A-4. Force Output for Triangular Membrane Element

A.5 ELEMENT NO. 5 (Plane Stress Quadrilateral)

The quadrilateral membrane element, shown in Figure A-5, is formed by placing four triangular membrane elements together. They are joined at the central node, v , which is taken to be the centroid of the quadrilateral. The triangular elements are those just described, hence the strain state in the quadrilateral is made up of four triangular regions of constant strain. Moreover, the general form of Hooke's Law is retained, allowing for anisotropic properties which are referred to reference axes specified by the angle β .

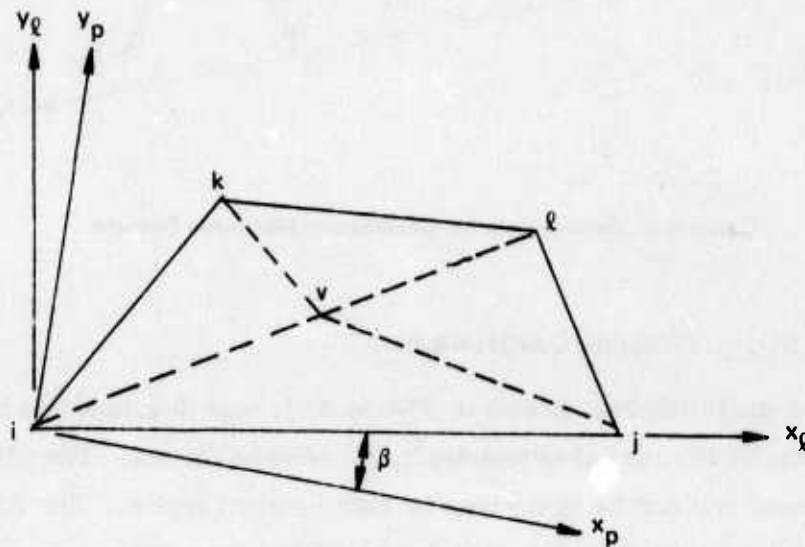


Figure A-5. Quadrilateral Membrane Element

The stiffness matrix of the quadrilateral is obtained by stacking the stiffnesses of the four triangles, and then eliminating the central node by requiring that no net forces exist there. The stress matrix gives the forces shown in Figure A-6 in terms of the nodal displacements. It should be noted that these forces do not constitute a nonredundant set of forces on the element, since the q 's are derived from the f 's. Details of the derivations are given in Reference 20.

Nodal stresses are determined with reference to local orthogonal axes, oriented so that at nodes i and j the x -axis is aligned with i - j and at nodes k and l it is aligned with k - l .

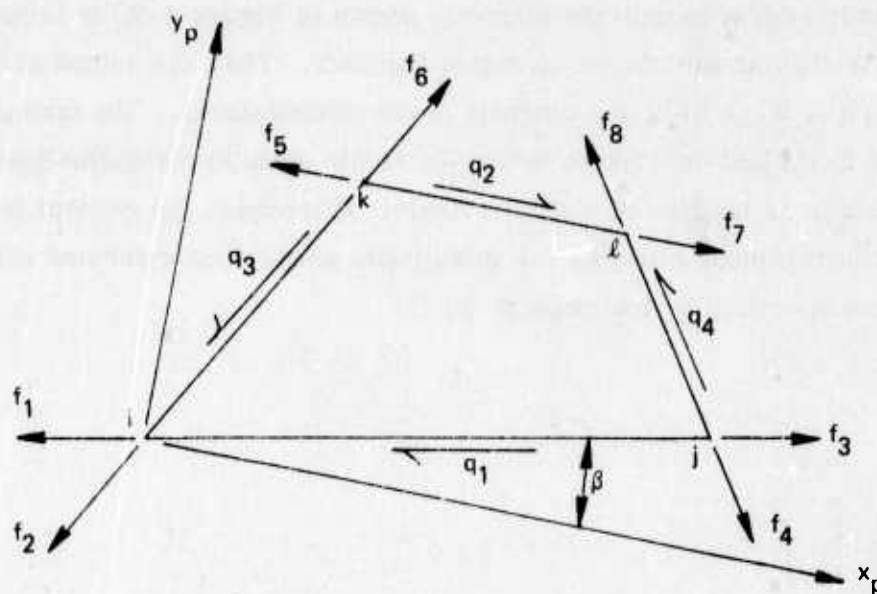


Figure A-6. Force Output for Quadrilateral Membrane Element

A.6 ELEMENT NO. 8 (Warped Quadrilateral)

The warped quadrilateral, shown in Figure A-7, was developed to idealize slightly-curved surfaces, loaded primarily by membrane forces. The element can be planar or warped and can be described by four or eight nodes. The first four nodes (i, j, k, and l) define the element, and the additional four nodes (m, n, o, and p) which are optional, are used to define the directions of the "kick" forces, k_1 , k_2 , k_3 , and k_4 , as shown in Figure A-8. When m, n, o, and p are not defined, the direction of the kick force at each corner is taken to be normal to the plane defined by the two adjacent edges.

As in the case of Element No. 5, four membrane triangles are assembled to form the quadrilateral. Node v, which is common to the four triangles, is at the intersection of lines ab and cd, which connect the mid-points of the sides. The element stiffness matrix is formed using a reference coordinate system (x_r , y_r , z_r). The x_r , y_r plane is determined as the one whose normal vector (in the z_r direction) is the cross product of vectors (i-l) and (j-k) in Figure A-7. By summing the

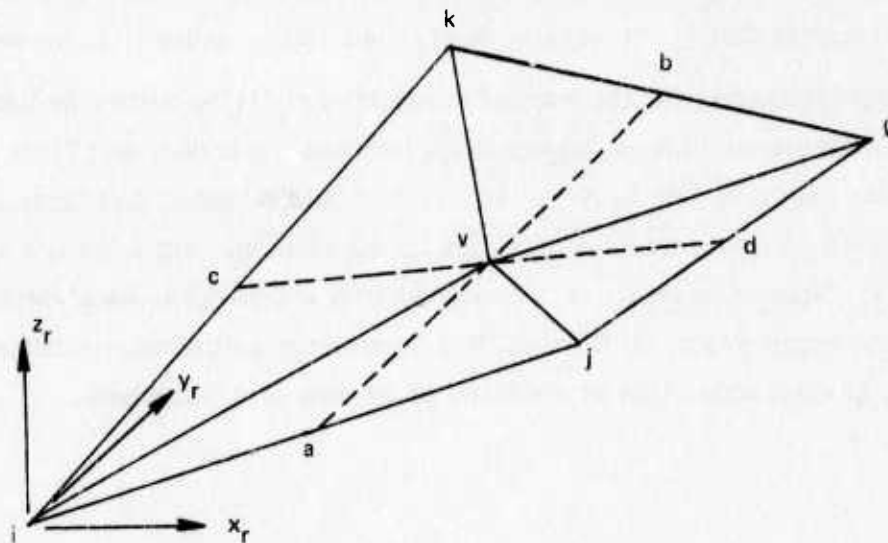


Figure A-7. Warped Quadrilateral Element

stiffnesses of the individual triangles, the following relationship is obtained in the reference coordinate system:

$$\{f\}_R = [k_q]_R \{\delta\}_R \quad (A.1)$$

where $[k_q]_R$ is the quadrilateral stiffness. Equation A.1 can be partitioned:

$$\begin{Bmatrix} f_e \\ f_i \end{Bmatrix} = \begin{bmatrix} k_{ee} & k_{ei} \\ k_{ie} & k_{ii} \end{bmatrix} \begin{Bmatrix} \delta_e \\ \delta_i \end{Bmatrix} \quad (A.2)$$

where subscripts e and i denote "external" and "internal", respectively, and:

$$\begin{aligned} \{f_i\} &= [f_{x_m} \quad f_{y_m} \quad f_{z_m}]^T \\ \{\delta_i\} &= [\delta_{x_m} \quad \delta_{y_m} \quad \delta_{z_m}]^T \end{aligned}$$

The stiffness matrix is reduced to a 12 x 12 matrix by imposing two conditions. First, it is required that no external forces be applied to node v in the x_r and y_r

direction ($f_{x_v} = f_{y_v} = 0$). The second condition is that lines c-v-d and a-v-b (Figure A-7) remain straight during deformation. It can be shown that this results in the requirement that f_{z_v} be equally distributed among nodes i, j, k, and l.

The stress matrix for the warped quadrilateral yields corner forces in the direction of the element edges, shear flows between the nodes, and "kick" forces at the nodes, as shown in Figure A-8. Again, it should be noted that these forces do not constitute a nonredundant set of forces on the element, since the q's are derived from the f's. Nodal stresses are determined with reference to local orthogonal axes, defined in the same way as in Element No. 5, with the additional specification that the y-axis, at each node, lies in the plane of the two adjacent edges.

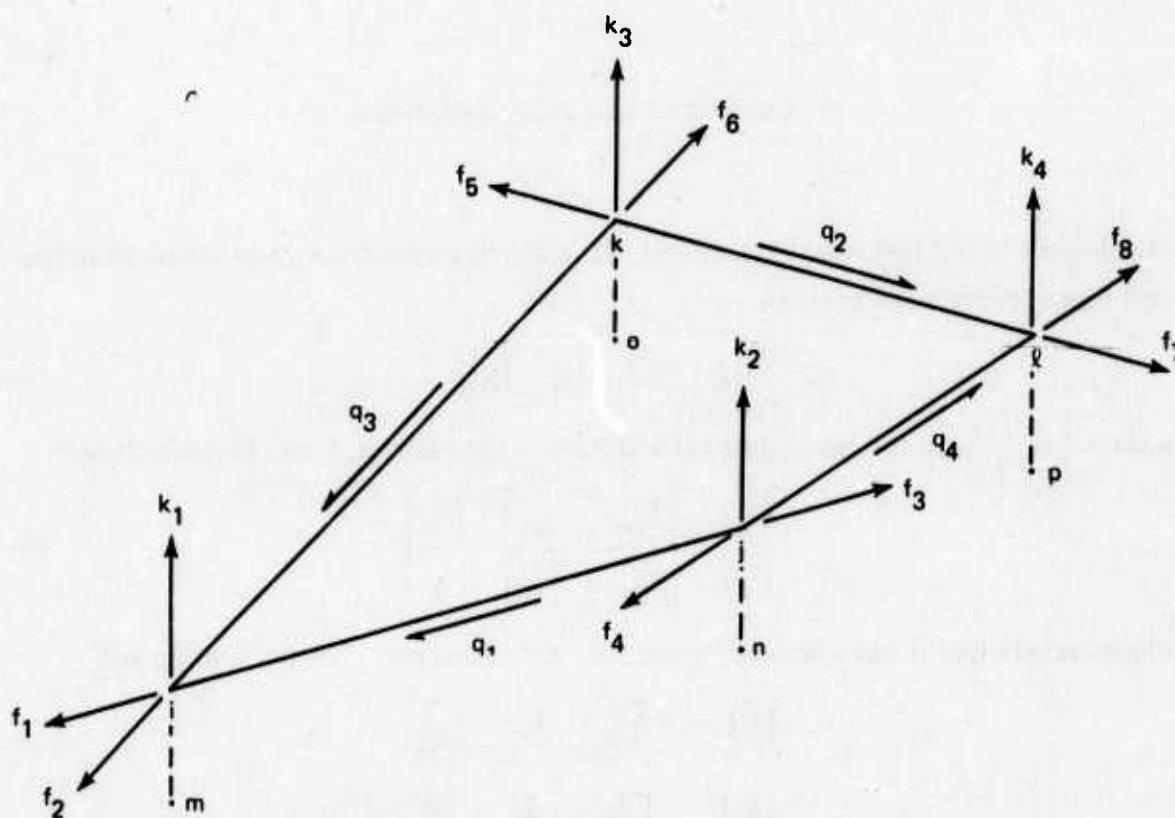


Figure A-8. Force Output for Warped Quadrilateral Element

A.7 ELEMENT NO. 6 (Warped Shear Panel)

As in the case of Element No. 8, this element can be planar or warped and can be described by four or eight nodes, with the second set of four nodes defining the "kick" force directions. The geometry and force output for this element are similar to that for Element No. 8, shown in Figures A-7 and A-8.

The element characteristics, for the warped-shear panel, are developed from the assumption of a distribution of stresses within the element that satisfy equilibrium but do not satisfy strain compatibility, except in the case of a parallelogram. A flexibility coefficient is then computed, using an energy formulation, and a set of equilibrium equations is used to obtain the stiffness matrix. Details involved in obtaining the energy expression can be found in Reference 7.

The flexibility coefficient is computed for the projected geometry of the warped quadrilateral on the reference plane defined by the lines 1-4 and 2-3. This coefficient, designated α , gives the relationship between the generalized shear deformation, δ , and the shear flow, q_{12} , acting along side 1-2, as shown in Figure A-9.

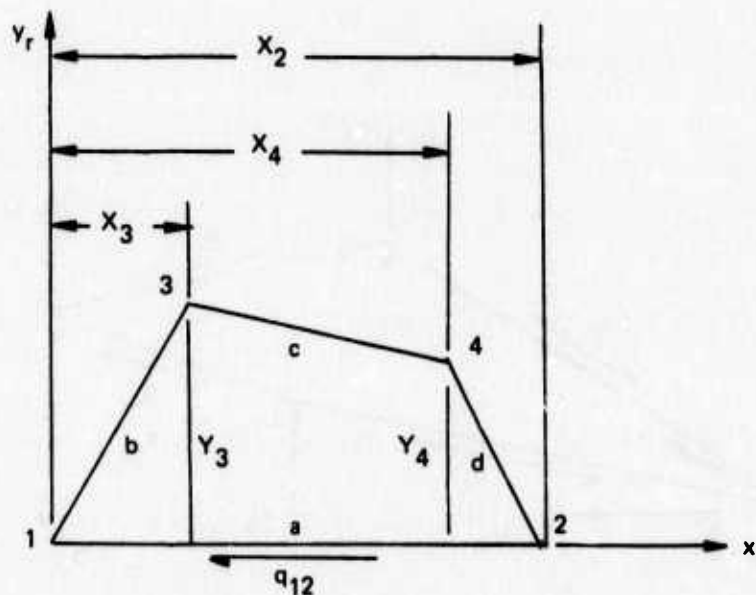


Figure A-9, Projected Quadrilateral

The computation of α for a particular quadrilateral depends on its shape, which can be any of the following types:

- (a) parallelogram or rectangle
- (b) trapezoid with side a parallel to side c
- (c) trapezoid with side b parallel to side d
- (d) general quadrilateral

In order to obtain the stiffness of the element, six equilibrium equations are written for the element in the reference coordinate system shown in Figure A-9.

The forces f'_1 , f'_2 , f'_3 , and f'_4 (Figure A-10) are necessary to insure equilibrium in the z-direction. The next step is to solve for f'_1 , f'_2 , f'_3 , f'_4 , q_2 , q_3 , and q_4 , in terms of q_1 . Since there are six equations and seven unknowns, an additional equation is needed, which is obtained by assuming that the resultant force in the z-direction passes through point v (Figure A-11), and that one-half of this resultant is acting at nodes j and k.

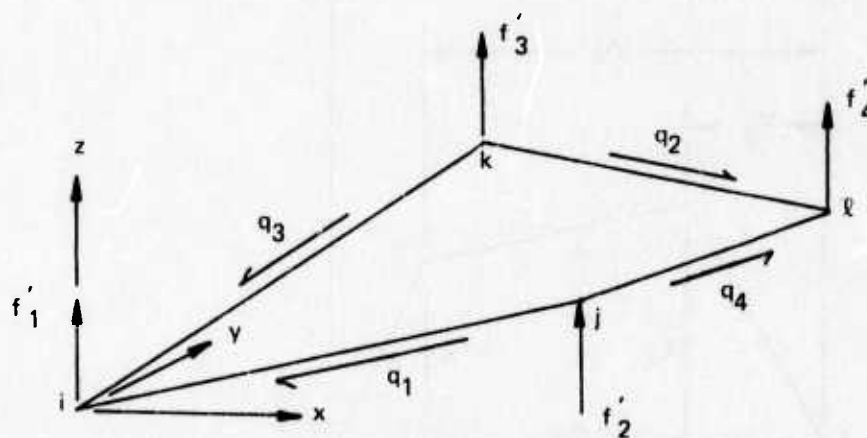


Figure A-10. Equilibrium of Warped Shear Panel

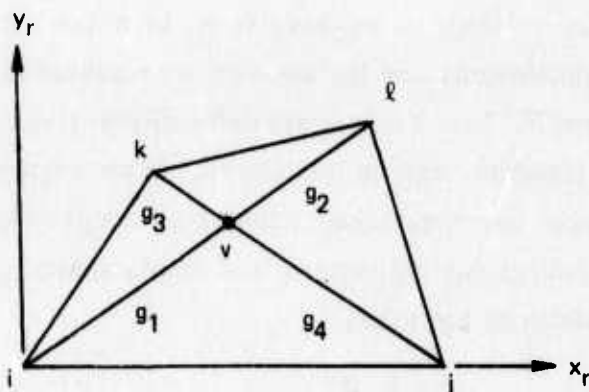


Figure A-11. Additional Geometry for Warped Shear Panel

Taking moments about a line passing through node j and parallel to i-l :

$$f'_3 = \frac{1}{2} \left(\frac{g_4}{g_3 + g_4} \right) (f'_1 + f'_2 + f'_3 + f'_4) \quad (\text{A.3})$$

or:

$$f'_1 + f'_2 + (1 - \gamma) f'_3 + f'_4 = 0 \quad (\text{A.4})$$

where:

$$\gamma = \frac{2(g_3 + g_4)}{g_4} \quad (\text{A.5})$$

The equilibrium equations and Equation (A.4) can be written in the matrix form:

$$[EQ] \{f''\} = \{R\} q_1 \quad (\text{A.6})$$

where $\{f''\} = \{f'_1 \ f'_2 \ f'_3 \ f'_4 \ q_2 \ q_3 \ q_4\}$. Solving Equation (A.6) we have:

$$\{f''\} = [EQ]^{-1} \{R\} q_1 \quad (\text{A.7})$$

or:

$$\{f\} = \{E\} q_1 \quad (\text{A.8})$$

where $\{f\}$ is obtained by enlarging $\{f'\}$ to include q_1 , and $\{E\}$ is obtained by correspondingly enlarging $[EQ]^{-1} \{R\}$ to include the relation $q_1 = q_1$.

The eight applied forces can be considered to correspond to eight degrees of freedom. They are related, as we have seen, by seven equations, six of which are based on static equilibrium, and the seventh is an assumed relationship. An additional relationship, based on elastic deformation and corresponding to a single elastic degree of freedom, can be introduced. If we regard q_1 as representing the generalized force corresponding to the elastic degree of freedom, and δ_{q_1} is defined as the corresponding generalized displacement, the elastic strain energy may be written in the form:

$$U = \frac{1}{2} \delta_{q_1} q_1 \quad (A.9)$$

Introducing the relationship:

$$\delta_{q_1} = \alpha q_1 \quad (A.10)$$

where α is a flexibility coefficient, mentioned earlier, we can write Equation (A.9) in the form:

$$U = \frac{1}{2\alpha} \delta_{q_1}^2 \quad (A.11)$$

An alternative form for the strain energy is:

$$U = \frac{1}{2} \{f\}^T \{\delta\} \quad (A.12)$$

where $\{\delta\}$ is a displacement matrix corresponding to $\{f\}$. Substitution of Equation (A.8) into Equation (A.12) yields:

$$U = \frac{1}{2} \{E\}^T q_1 \quad (A.13)$$

Comparing Equations (A.9) and (A.13), we see that:

$$\delta_{q_1} = \{E\}^T \{\delta\} \quad (A.14)$$

Substitution of Equation (A.14) into Equation (A.11) yields the form:

$$U = \frac{1}{2\alpha} \{\delta\}^T \{E\} \{E\}^T \{\delta\} \quad (A.15)$$

or:

$$U = \frac{1}{2} \{\delta\}^T [k] \{\delta\} \quad (A.16)$$

where:

$$[k] = \frac{1}{\alpha} \{E\} \{E\}^T \quad (A.17)$$

$[k]$ is seen to be the stiffness matrix in the relationship:

$$\{f\} = [k] \{\delta\} \quad (A.18)$$

As the program can only accept element stiffness matrices in a form that relates element nodal forces to nodal displacement, there is a transformation of $[k]$ to such a form.

The stress matrix for the element is similar to that of Element No. 8, in that it gives the corner forces, "kick" forces, and shear flows.

A.8 ELEMENT NO. 11 (Hinged beam)

This element is similar to Element No. 2, except that: 1) it is hinged on the z_j axis at node j (see Figure 24), and 2) there is no provision for offset of the centroidal axis from the nodes defining the points of attachment to adjoining structure.

Appendix B

INPUT DATA FOR FINITE ELEMENTS

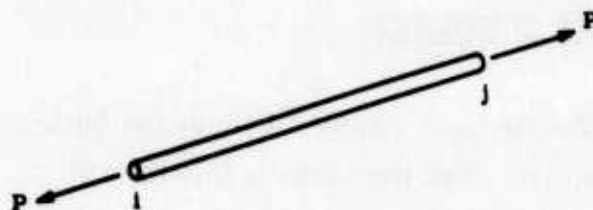
The following pages illustrate and describe the input member data for the finite elements that are currently in the ASOP-3 program. The input data is filled out in fields in accordance with Figure 20 and Subsection 4.2.6. It should be noted that it is not necessary to provide cards for the elastic properties of each individual member (data class 2), if the appropriate data is entered through the material properties input. In any case, a material code must be provided. As the data class 5, subclass 1 and 2 cards, containing allowable stresses and minimum and maximum gages, are in the same format for all the elements (Figure 21), they are omitted in the following pages. Only topology, elastic property information, and corner force output are shown.

The column matrices labeled "OUTPUT" contain the element nodal forces. These forces, including edge shear flows in some cases, are printed out optionally.

Special care should be exercised in entering the angle, β , which is used when property axes differ from element axes. This angle is always measured from the element x_l -axis, which is along the i-j side, with origin at i. It is always positive in the direction away from the element. It should also be noted that, in numbering the nodes of the quadrilateral elements, i-j need not be in the counterclockwise direction. However, k must be on a common edge with i, and l with j.

In the case of the beam elements (Nos. 2 and 11), the only cross-sectional properties listed are the area, the moments of inertia I_{yy} and I_{zz} , and the effective polar moment of inertia J (used in determining torsional stiffness). No cross-sectional dimensions are explicitly introduced, as the assumption is made that the whole cross-sectional area is concentrated at the location of the extreme fibers, which can be determined from the area and the moments of inertia, and is so determined in the program. Care should be exercised in defining the angle β , which establishes the orientation of the y and z axes for beam elements, as bending moments about the y-axis are disregarded in resizing. The xy plane should be the plane in which the element is primarily loaded in bending.

It should also be noted that only Element No. 2 can be treated as an offset beam. Element No. 11 does not have this capability.

Bar Element**ELEMENT No. 1****INPUT**

Member No.	Member Type	Matl. Code		Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No. 1	1				1	1	i	j			Area				
* No.					2	1					E				

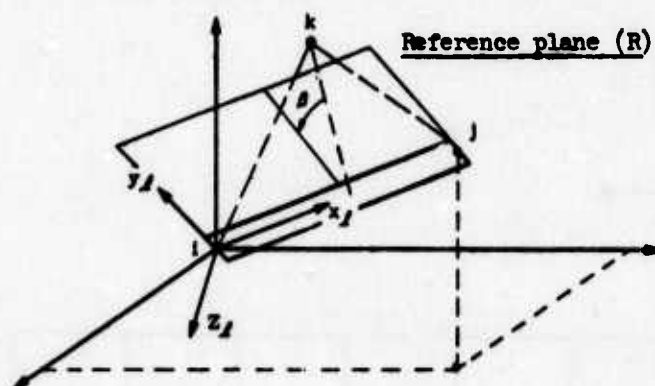
E = Modulus of Elasticity

*This card needed only when overriding properties in the materials data.

OUTPUT

P (Axial Load)

Beam Element



ELEMENT No. 2

Nodes i, j, k determine reference plane R. Angle β determines orientation of y_1, z_1 axes with respect to reference plane R.

For Offset Beam: Nodes i, j, are the structural connection nodes and nodes l, m are the beam centroid nodes. Node k determines the orientation of the beam.

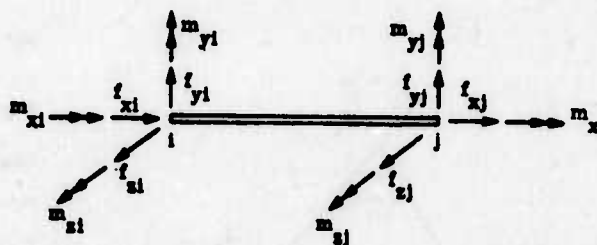
INPUT

Member No.	Member Type	Matl. Code	Graphics	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No.	2				1	1	i	j	k	l	Area	β	I_{yy}	I_{zz}	J
** No.					1	2	m								
* No.					2	1					E				

OUTPUT

β is in degrees
J = Effective polar moment of inertia
E = Modulus of Elasticity

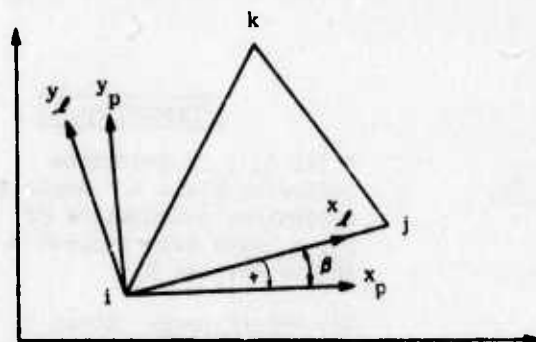
$$\left\{ \begin{array}{l} f_{x1} \\ f_{y1} \\ f_{z1} \\ m_{x1} \\ m_{y1} \\ m_{z1} \\ f_{xj} \\ f_{yj} \\ f_{zj} \\ m_{xj} \\ m_{yj} \\ m_{zj} \end{array} \right\} \text{ in local system}$$



*This card needed only when overriding properties in the materials data.
**This card needed only for the offset beam.

Triangular Membrane Element

ELEMENT No. 4



x_l, y_l - Local Axes

x_p, y_p - Local Property Axes

INPUT

Isotropic and Orthotropic

Member No.	Member Type	Matl. Code	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No. 4				1	1	i	j	k		t		β		
* No.				2	1					A_{11}	A_{22}	A_{33}	A_{12}	

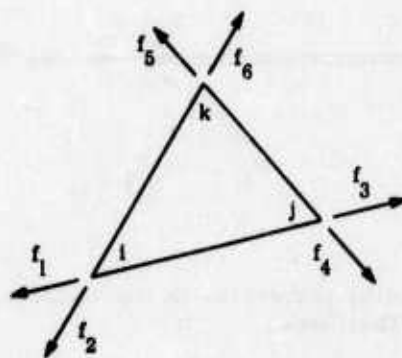
Anisotropic

No. 4				1	1	i	j	k		t		β		
* No.				2	1					A_{11}	A_{22}	A_{33}	A_{12}	A_{23}
* No.				2	2					A_{13}				

*This card needed only when overriding properties in the materials data.

OUTPUT

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix}$$



Note:

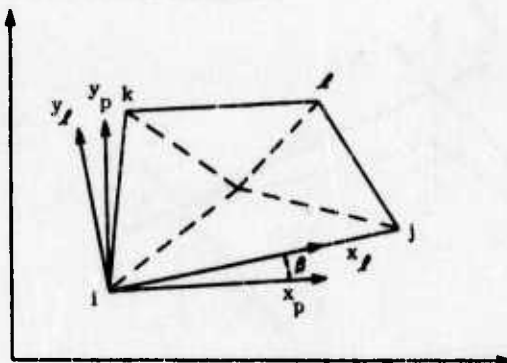
t = thickness of element
 β given in degrees

Elastic factors (A_{11} etc.) are
elements of stress-strain law:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{21} & A_{22} & A_{23} \\ A_{31} & A_{32} & A_{33} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

Quadrilateral Membrane Element

ELEMENT No. 5



Element No. 5 is composed of four triangular elements.

x_l, y_l - Local axes

x_p, y_p - Local property axes

INPUT

Isotropic and Orthotropic

Member No.	Member Type	Matl. Code	Graphics	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No. 5					1	1	i	j	k	l	t	β			
* No.					2	1					A_{11}	A_{22}	A_{33}	A_{12}	

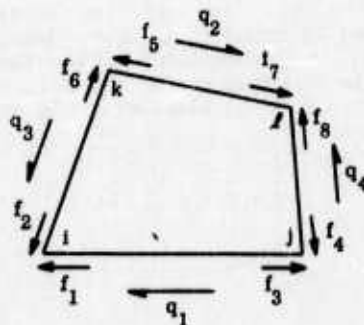
Anisotropic

No. 5					1	1	i	j	k	l	t	β			
* No.					2	1					A_{11}	A_{22}	A_{33}	A_{12}	A_{23}
* No.					2	2					A_{13}				

*This card needed only when overriding properties in the materials data.

OUTPUT

$\left\{ \begin{array}{l} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \\ f_7 \\ f_8 \\ q_1 \\ q_2 \\ q_3 \\ q_4 \end{array} \right\}$



Note:

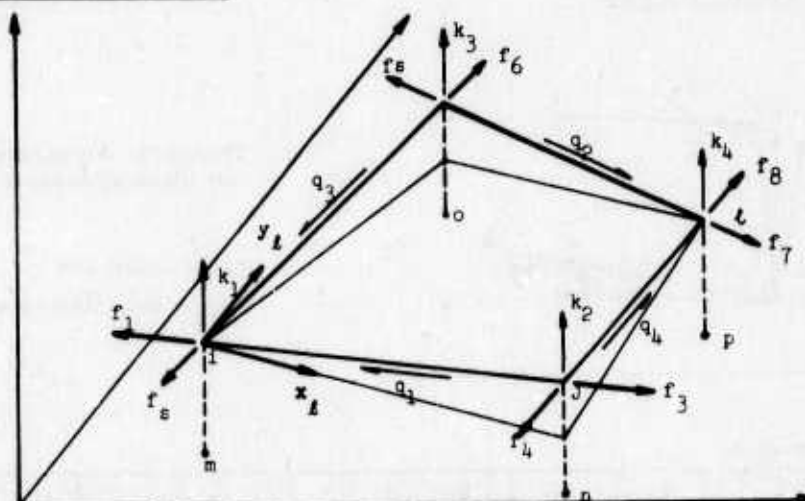
t = thickness of element
 β given in degrees

Elastic factors (A_{11} etc.) are elements of stress-strain law:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{21} & A_{22} & A_{23} \\ A_{31} & A_{32} & A_{33} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

Quadrilateral Shear Panel (Garvey)

ELEMENT No. 6



Panel can be warped or planar

INPUT

Member No.	Member Type	Matl. Code	Graphics	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No. 6					1	1	i	j	k	l	t				
** No.					1	2	m	n	o	p					
* No.					2	1					E	G			

*This card needed only when overriding properties in the materials data.

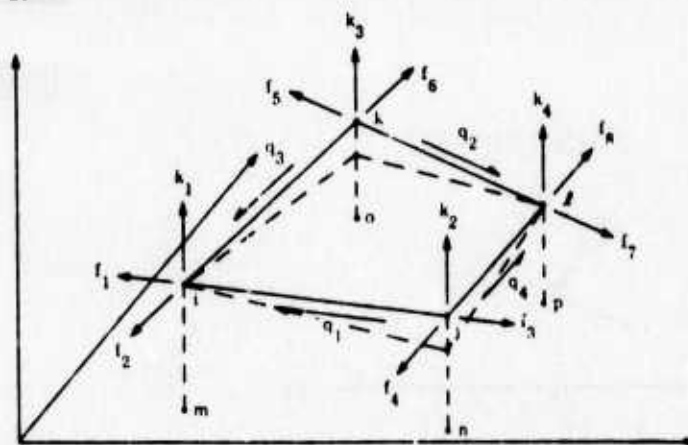
t = thickness of panel
E = modulus of elasticity
G = shear modulus

OUTPUT

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \\ f_7 \\ f_8 \\ q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix} \quad \begin{Bmatrix} k_1 \\ k_2 \\ k_3 \\ k_4 \end{Bmatrix}$$

Note:

** Nodes m, n, o, and p are optional (data card 12). They are used to specify the directions of the "kick" forces. When nodes m, n, o, and p are specified the direction of the "kick" forces is from i to m, from j to n, from k to o, and from l to p. When these nodes are not specified (card 12 is left out), the direction of the "kick" forces is perpendicular to the two adjacent sides at a node and its sense is as shown above.



INPUT

Isotropic and Orthotropic

Member No.	Member Type	Matl. Code	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No.	8			1	1	i	j	k	l	t	θ			
* No.				1	2	m	n	o	p					
** No.				2	1					A_{11}	A_{22}	A_{33}	A_{12}	

Anisotropic

No.	8			1	1	i	j	k	l	t	θ			
* No.				1	2	m	n	o	p					
** No.				2	1					A_{11}	A_{22}	A_{33}	A_{12}	A_{23}
** No.				2	2					A_{13}				

OUTPUT

Note:

*Nodes m, n, o, p are optional (data card 12). They are used to specify the directions of the "Kick" forces. If data card 12 is left out, direction of "Kick" forces is perpendicular to adjacent sides at node and in the directions shown in the figure.

t = Thickness of element

θ = Angle between property axes and side i-j; given in degrees. Elastic factors (A_{11} etc.) are elements of stress strain law

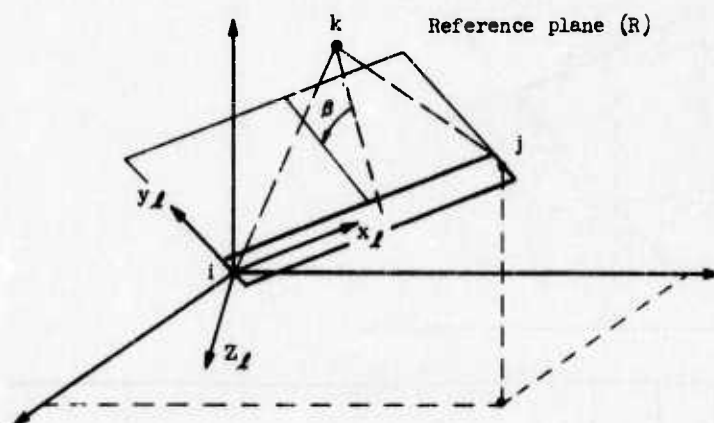
$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \\ f_7 \\ f_8 \end{Bmatrix} \quad \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ k_1 \\ k_2 \\ k_3 \\ k_4 \end{Bmatrix}$$

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{21} & A_{22} & A_{23} \\ A_{31} & A_{32} & A_{33} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

**This card needed only when overriding properties in the materials data.

Beam Element

ELEMENT No. 11



INPUT

Member No.	Member Type	Matl. Code	Graphics	Const. Code	Data Class	Sub Class	Node 1	Node 2	Node 3	Node 4	Factor 1	Factor 2	Factor 3	Factor 4	Factor 5
No.	2				1	1	i	j	k		Area	β	I_{yy}	I_{zz}	J
* No.					2	1					E				

β is in degrees

J = Effective polar moment of inertia

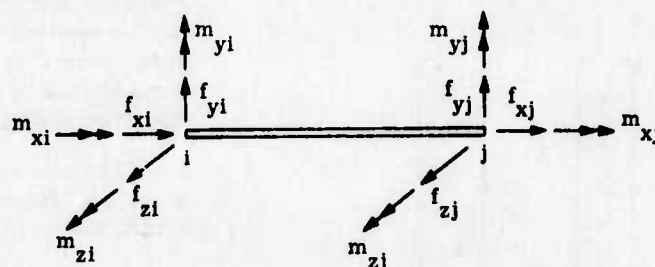
E = Modulus of Elasticity

*This card needed only when overriding properties in the materials data.

OUTPUT

$$\begin{Bmatrix} f_{xi} \\ f_{yi} \\ f_{zi} \\ m_{xi} \\ m_{yi} \\ m_{zi} \\ f_{xj} \\ f_{yj} \\ f_{zj} \\ m_{xj} \\ m_{yj} \\ m_{zj} \end{Bmatrix}$$
 in local system

m_{zj} should be zero



Sign convention for moments depends on whether right or left-hand coordinate system is used. For right-hand system, right-hand rule holds, for left-hand system, left-hand rule holds.

Appendix C

COMPUTATION OF ANGLE BETWEEN LOCAL ELEMENT AXES AND PROPERTY AXES

The angle β is defined as the angle between the local element x-axis and the property x-axis, positive away from the element, as shown in Figure C-1. In the case of triangular elements and planar quadrilateral elements, the property axes lie in the same plane as the local element axes, which is the plane in which the element lies. In the case of warped quadrilateral elements, it is necessary to define an element plane. That plane is chosen here to be the i-j-k plane, and, again, the local element axes and property axes, and consequently the angle β , all lie in it.

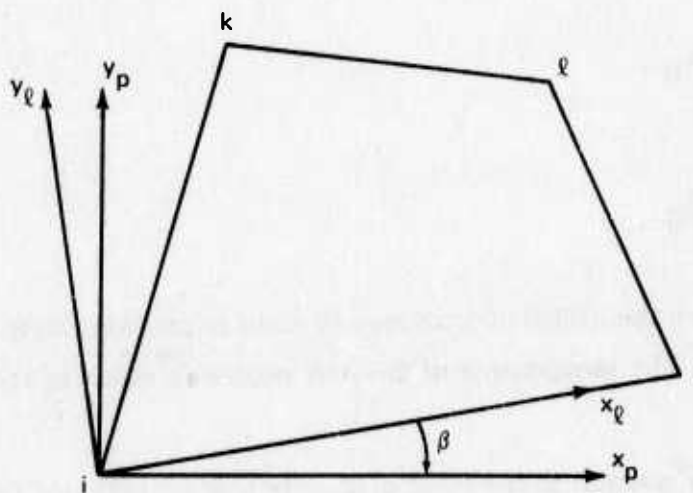


Figure C-1. Definition of Angle β

The reference direction for the property x-axis is defined by means of the unit vector \overrightarrow{AB} , along the line AB, where A and B are two points defined in the global axis system. The components of \overrightarrow{AB} in the global axis system are given by:

$$d_{ab_x} = (x_b - x_a)/L_{ab} \quad (C. 1)$$

$$d_{ab_y} = (y_b - y_a)/L_{ab}$$

$$d_{ab_z} = (z_b - z_a)/L_{ab}$$

where x_a , y_a , and z_a are the global coordinates of point A, and similarly for point B, and L_{ab} is the length of AB, given by:

$$L_{ab} = \left[(x_b - x_a)^2 + (y_b - y_a)^2 + (z_b - z_a)^2 \right] \quad (C.2)$$

The components of the unit vector \vec{ij} along ij are given by:

$$d_{ij_x} = (x_j - x_i)/L_{ij}$$

$$d_{ij_y} = (y_j - y_i)/L_{ij} \quad (C.3)$$

$$d_{ij_z} = (z_j - z_i)/L_{ij}$$

where x_i , y_i , and z_i are the global coordinates of node i, and similarly for node j, and L_{ij} is the length of ij . The components of the unit vector \vec{ik} along ik are similarly formed.

The unit vector \vec{w} normal to the i - j - k plane can now be obtained from the relation:

$$\vec{w} = (\vec{ij} \times \vec{ik})/L_w \quad (C.4)$$

where L_w is the length of the vector formed by the vector product in the numerator.

The components of these vectors are:

$$d_{w_x} = (d_{ij_y} d_{ik_z} - d_{ij_z} d_{ik_y})/L_w \quad (C.5)$$

$$d_{w_y} = (d_{ij_z} d_{ik_x} - d_{ij_x} d_{ik_z}) / L_w \quad (C.5)$$

$$d_{w_z} = (d_{ij_x} d_{ik_y} - d_{ij_y} d_{ik_x}) / L_w$$

and L_w is the square root of the sum of the squares of the numerators in the above expressions.

The projection of \overrightarrow{AB} on the i-j-k plane is obtained by subtracting from \overrightarrow{AB} the component of \overrightarrow{AB} along \overrightarrow{w} . A unit vector \overrightarrow{b} along that projection is then given by:

$$\overrightarrow{b} = [\overrightarrow{AB} - (\overrightarrow{AB} \cdot \overrightarrow{w}) \overrightarrow{w}] / L_b \quad (C.6)$$

where the components of \overrightarrow{b} are:

$$d_{b_x} = (d_{ab_x} - S d_{w_x}) / L_b$$

$$d_{b_y} = (d_{ab_y} - S d_{w_y}) / L_b$$

$$d_{b_z} = (d_{ab_z} - S d_{w_z}) / L_b$$

(C.7)

and L_b is the square root of the sum of the squares of the numerators in the above expressions, while:

$$S = \overrightarrow{AB} \cdot \overrightarrow{w} = d_{ab_x} d_{w_x} + d_{ab_y} d_{w_y} + d_{ab_z} d_{w_z} \quad (C.8)$$

The angle β is now determined from:

$$\begin{aligned} \cos \beta &= \overrightarrow{b} \cdot \overrightarrow{ij} \\ &= d_{b_x} d_{ij_x} + d_{b_y} d_{ij_y} + d_{b_z} d_{ij_z} \end{aligned} \quad (C.9)$$

The algebraic sign of β is determined by comparing the directions of $\vec{i} \vec{j} \times \vec{i} \vec{k}$ (or \vec{w}) and $\vec{b} \times \vec{i} \vec{j}$, which are colinear. If they are in the same direction, β is positive. If they are in opposite directions, β is negative. Their relative directions are determined by forming the scalar product of the two vectors. That is, the sign of β is given by the sign of $\vec{w} \cdot (\vec{b} \times \vec{i} \vec{j})$.

Appendix D

SUPPLEMENTARY MATHEMATICAL DETAILS PERTAINING TO STRUCTURAL ANALYSIS

D.1 MODIFIED CHOLESKY ALGORITHM

The possibility of decomposing the stiffness matrix in the following manner:

$$[K] = [L] [L]^T \quad (D-1)$$

where $[L]$ is a lower triangular matrix, is guaranteed if K is positive definite. The force-displacement relations then become:

$$[L][L]^T \{\Delta\} = \{F\} \quad (D-2)$$

Once L is obtained by Cholesky decomposition (Reference 20), use is made of the equation:

$$[L] \{Z\} = \{F\} \quad (D-3)$$

for generating the elements of the vector $\{Z\}$ in succession (i. e. , top to bottom).

This step is then followed by a backward substitution in the relationship:

$$[L]^T \{\Delta\} = \{Z\} \quad (D-4)$$

to determine the elements of $\{\Delta\}$ in reverse order.

The standard Cholesky formulae to determine the elements of the matrix $[L]$ are:

$$l_{kk} = \left(a_{kk} - \sum_{j=1}^{k-1} l_{kj}^2 \right)^{1/2} \quad (D-5a)$$

for the diagonal elements, and:

$$l_{ik} = \left(a_{ik} - \sum_{j=1}^{k-1} l_{ij} l_{kj} \right) / l_{kk} \quad (D-5b)$$

for $i > k$, where a_{ij} are the elements of $[K]$. However, since the matrices which normally occur in the solution of large practical structural problems contain a large number of zero entries, the present solution scheme seeks to benefit from the presence of the zero elements, by modifying the above equations to read:

$$l_{kk} = \left(a_{kk} - \sum_{j=p(k)}^{k-1} l_{kj}^2 \right)^{1/2} \quad (D-6a)$$

for the diagonal elements, and, for $i > k \geq p(i)$:

$$l_{ik} = \left(a_{ik} - \sum_{j=q(i,k)}^{k-1} l_{ij} l_{kj} \right) / l_{kk} \quad (D-6b)$$

with $l_{ik} = 0$ for $1 \leq k < p(i)$, where $p(i)$ designates the position of the first nonzero element of the i^{th} row of the matrix $[K]$, and $q(i, k)$ denotes the larger of the numbers $(p(i), p(k))$. The modified formulae, Equations (D. 6), enable the suppression of many zero terms and, for banded matrices (those with all nonzero terms near the main diagonal), this saving may reduce the volume of computation by a large factor. The modified procedure also facilitates computer storage economy and input-output operations of highly-banded matrices, through condensation of the $[K]$ and $[L]$ matrices.

D.2 COORDINATE TRANSFORMATIONS

It is frequently necessary in the program to transform stresses, strains, and stiffness coefficients from local element axes to property axes, or vice versa. Accordingly, the required transformation matrices are now defined.

A typical element is shown in Figure D-1 with its local element axes, x_ℓ , y_ℓ , and z_ℓ , and property axes, x_p , y_p , and z_p . The angle between these sets of axes is shown in the positive sense in Figure D-1 (a), that is, positive away from the

element. In that figure, this is seen to represent a clockwise rotation from the local-element axes to the property axes. However, it should be carefully noted that this rotation would be counterclockwise for positive β , if the nodes were ordered so that i and j were interchanged, as seen in Figure 39 (b), where the same effective orientation of the property axes results in an equal but negative value of β .

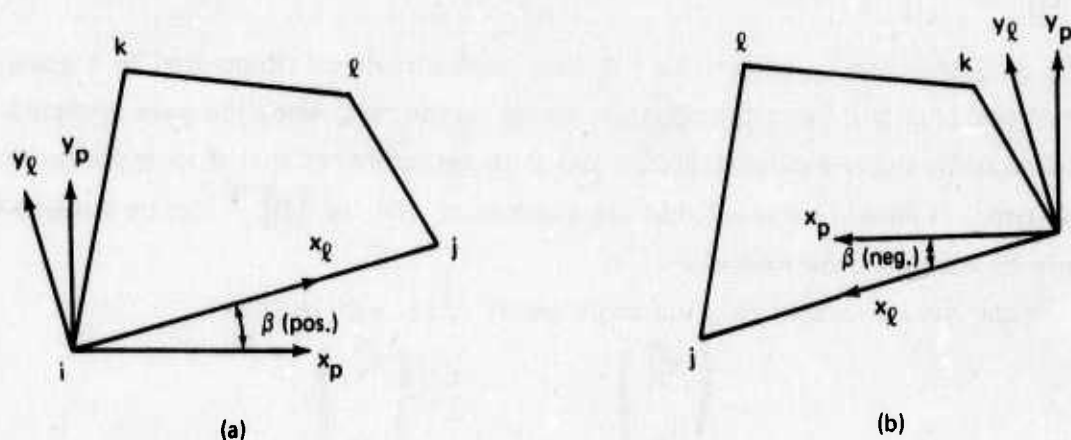


Figure D-1. Element Axis Systems

The transformation relations for stress and strain in a rotation from the local element axes to the property axes are written as (Reference 19, p. 19):

$$\begin{Bmatrix} \sigma_x^p \\ \sigma_y^p \\ \tau_{xy}^p \end{Bmatrix} = [B] \begin{Bmatrix} \sigma_x^l \\ \sigma_y^l \\ \tau_{xy}^l \end{Bmatrix} \quad (D-7)$$

and

$$\begin{Bmatrix} \epsilon_x^p \\ \epsilon_y^p \\ \gamma_{xy}^p \end{Bmatrix} = [D] \begin{Bmatrix} \epsilon_x^l \\ \epsilon_y^l \\ \gamma_{xy}^l \end{Bmatrix} \quad (D-8)$$

where

$$[D] = \begin{bmatrix} \cos^2 \beta & \sin^2 \beta & -\sin \beta \cos \beta \\ \sin^2 \beta & \cos^2 \beta & \sin \beta \cos \beta \\ 2 \sin \beta \cos \beta & -2 \sin \beta \cos \beta & \cos^2 \beta - \sin^2 \beta \end{bmatrix} \quad (D-9)$$

and $[B] = ([D]^T)^{-1}$

These relations can be seen to be valid for both situations illustrated in Figure D-1, as the rotations will be in the opposite sense in the two, when the axis systems are all taken to be right-handed systems and it is remembered that β is opposite in sign in the two. It should be noted that the inverse of $[D]$ or $[D]^T$ can be obtained simply by changing the sign of β .

If the stress-strain relation in property axes is given by:

$$\begin{Bmatrix} \sigma_x^p \\ \sigma_y^p \\ \tau_{xy}^p \end{Bmatrix} = [A] \begin{Bmatrix} \epsilon_x^p \\ \epsilon_y^p \\ \gamma_{xy}^p \end{Bmatrix} \quad (D-10)$$

it can be seen that substitution of Equations (D.7) and (D.8) into Equation (D.10) yields:

$$\begin{Bmatrix} \sigma_x^l \\ \sigma_y^l \\ \tau_{xy}^l \end{Bmatrix} = [\bar{A}] \begin{Bmatrix} \epsilon_x^l \\ \epsilon_y^l \\ \gamma_{xy}^l \end{Bmatrix} \quad (D-11)$$

where:

$$[\bar{A}] = [D]^T [A] [D] \quad (D-12)$$

D.3 STIFFNESS PROPERTIES OF COMPOSITE ELEMENTS

As discussed in Subsection 2.1, layers of composite elements are treated internally in the program as separate elements in the analysis of the structure for nodal deflections and internal loads.

The stress-strain relation for a layer of the composite in fiber axes (x-axis in fiber direction) is given by (Reference 19, p. 18):

$$\begin{Bmatrix} \sigma_x^f \\ \sigma_y^f \\ \tau_{xy}^f \end{Bmatrix} = [Q] \begin{Bmatrix} \epsilon_x^f \\ \epsilon_y^f \\ \gamma_{xy}^f \end{Bmatrix} \quad (D-13)$$

where:

$$[Q] = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{21} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}$$

and:

$$Q_{11} = E_{11}/(1 - \nu_{12}\nu_{21})$$

$$Q_{22} = E_{22}/(1 - \nu_{12}\nu_{21})$$

$$Q_{12} = Q_{21} = \nu_{21}E_{11}/(1 - \nu_{12}\nu_{21}) = \nu_{12}E_{22}/(1 - \nu_{12}\nu_{21})$$

$$Q_{66} = G_{12}$$

with E_{11} and E_{22} being the Young's moduli along and transverse to the fiber direction, respectively, G_{12} the shear modulus and ν_{12} the Poisson's ratio for transverse strain due to stress along the fibers (and vice versa for ν_{21}). It should be noted that, for the purpose of analyzing the structure to determine nodal displacements and internal loads, these stiffness parameters include the stiffness of the matrix material, which is presumed to remain intact.

In forming the element stiffness matrix, relating nodal forces to nodal displacements for each layer treated as a separate element, it is necessary first to determine the stress-strain relation for the layer material in local element axes, as shown in Figure D-2.

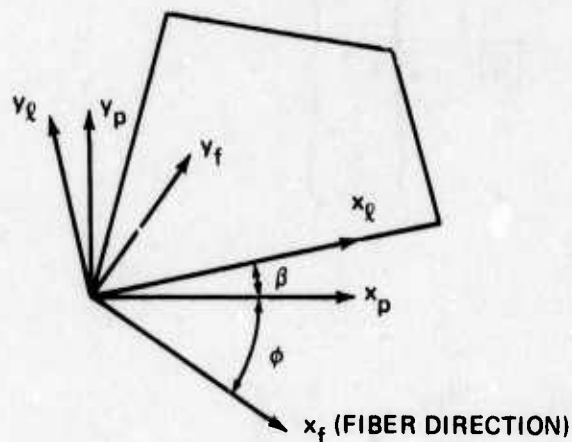


Figure D-2. Axis Systems for Layers of a Composite Element

This relation is obtained from Equation (D-13) by a coordinate transformation, and is given by:

$$\begin{Bmatrix} \sigma_x^l \\ \sigma_y^l \\ \tau_{xy}^l \end{Bmatrix} = [\bar{A}] \begin{Bmatrix} \epsilon_x^l \\ \epsilon_y^l \\ \gamma_{xy}^l \end{Bmatrix} \quad (D-14)$$

where:

$$[\bar{A}] = [D]^T [Q] [D]$$

and $[D]$ is as defined in Equation (D-9), with $\beta + \phi$ replacing β , as the rotation is now through the angle $\beta + \phi$ rather than β . This relation is then used in the membrane element subroutines (types 4, 5, and 8) to determine the required element stiffness matrix.

Appendix E

DETERMINATION OF AVERAGE STRESS RESULTANTS IN TRIANGULAR AND QUADRILATERAL ELEMENTS

E.1 DETERMINATION OF AVERAGE STRESS RESULTANTS IN TRIANGULAR ELEMENTS

The internal forces on triangular elements generated in ASOP-3 are corner forces in the directions of the element edges, as shown in Figure E-1. It is necessary to transform these forces into stress resultants that are consistent with the uniform strain presumed to exist in the element.

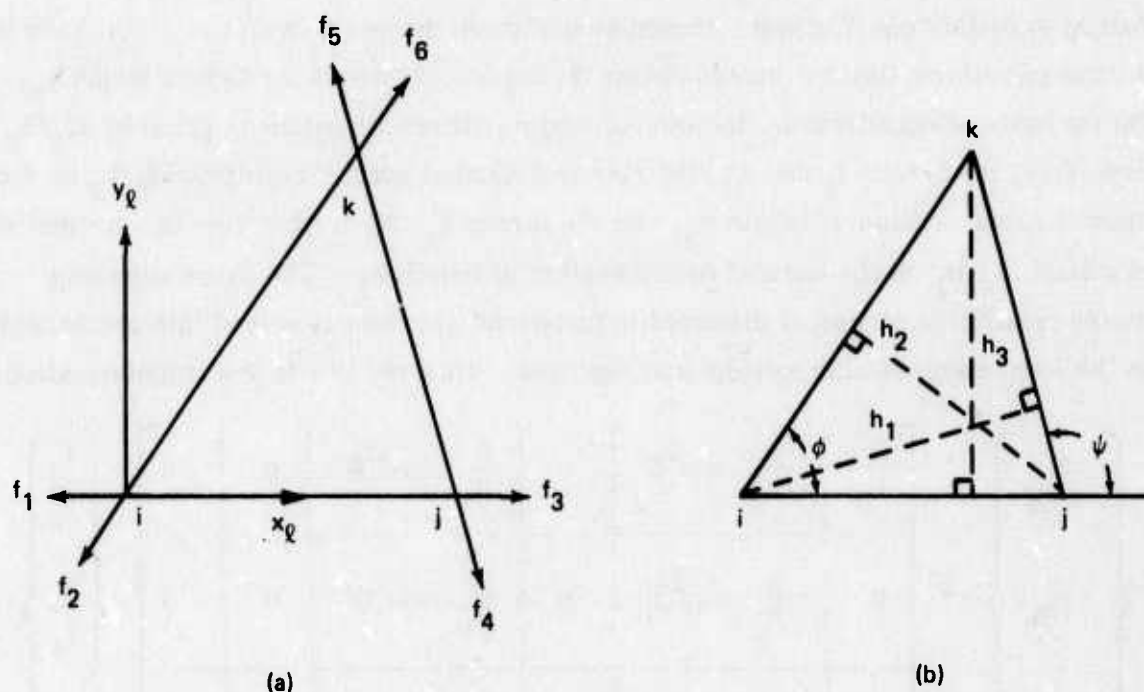


Figure E-1. Triangular Element Geometry and Corner Forces

In developing this transformation, it is necessary to determine the angles, ϕ and ψ , and the lengths, h_1, h_2 , and h_3 , of the perpendiculars from the three vertices to the opposite sides. We start by determining the lengths, L_{ij} , L_{ik} , and L_{jk} , of the three edges, ij , ik , and jk , and the components of unit vectors, \vec{ij} , \vec{ik} ,

and jk , along these edges, as shown in Appendix C. The angles, ϕ and ψ , are then determined from the relations:

$$\begin{aligned}\cos \phi &= \frac{\vec{ij} \cdot \vec{ik}}{L_{ij} L_{ik}} \\ \cos \psi &= \frac{\vec{ij} \cdot \vec{jk}}{L_{ij} L_{jk}}\end{aligned}\quad (E-1)$$

and the perpendicular lengths are determined from the relations:

$$h_1 = \frac{2A}{L_{jk}}, \quad h_2 = \frac{2A}{L_{ik}}, \quad h_3 = \frac{2A}{L_{ij}} \quad (E-2)$$

where A is the area of the triangular, and is given by:

$$A = \frac{1}{2} \left| \vec{ij} \times \vec{ik} \right| L_{ij} L_{ik} \quad (E-3)$$

In effecting the transformation from corner forces to stress resultants, we consider the three pairs of colinear forces, which are necessarily equal and opposite forces to satisfy equilibrium. Associated with the forces f_1 and f_3 ($= f_1$) there is a state of uniform normal stress acting on the normal cross section of length h_3 . On the basis of equilibrium, the corresponding stress resultant is given by $2f_1/h_3$. Similarly, the forces f_2 and f_6 give rise to a normal stress resultant $2f_2/h_2$ on the normal cross section of length h_2 , and the forces f_4 and f_5 give rise to a normal stress resultant $2f_4/h_1$ on the normal cross section of length h_1 . The three resulting states of uniform stress at different orientations are then resolved into components in the local element axis system and summed. This results in the transformation:

$$\begin{Bmatrix} N_{x_\ell} \\ N_{y_\ell} \\ N_{xy_\ell} \end{Bmatrix} = \begin{bmatrix} \frac{2}{h_3} & \frac{2}{h_2} \cos^2 \phi & 0 & \frac{2}{h_1} \cos^2 \psi & 0 & 0 \\ 0 & \frac{2}{h_2} \sin^2 \phi & 0 & \frac{2}{h_1} \sin^2 \psi & 0 & 0 \\ 0 & \frac{1}{h_2} \sin 2\phi & 0 & \frac{1}{h_1} \sin 2\psi & 0 & 0 \end{bmatrix} \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix}$$

If the stress resultants in the property axis system are desired, the following transformation is carried out:

$$\begin{Bmatrix} N_{x_p} \\ N_{y_p} \\ N_{xy_p} \end{Bmatrix} = [B] \begin{Bmatrix} N_{x_\ell} \\ N_{y_\ell} \\ N_{xy_\ell} \end{Bmatrix}$$

where the transformation matrix $[B]$ is as defined in Equation (D-7).

E.2 DETERMINATION OF AVERAGE STRESS RESULTANTS IN QUADRILATERAL ELEMENTS.

In determining the average stress in a quadrilateral element, the element is cut successively along the two lines joining the midpoints of opposite edges and, in each case, forces acting on the cut section are determined by putting the free body on one side of the cut in equilibrium. This is illustrated in Figures E-2 and E-3.

Figure E-2 shows the element with the corner forces acting on it in the directions of the element edges. Points a, b, c, and d are the midpoints of the edges, and the directions of the lines cd and ab define oblique coordinate axes u and v. A third coordinate axis w, normal to u and v, completes the triad to form the right-handed axis system u, v, w.

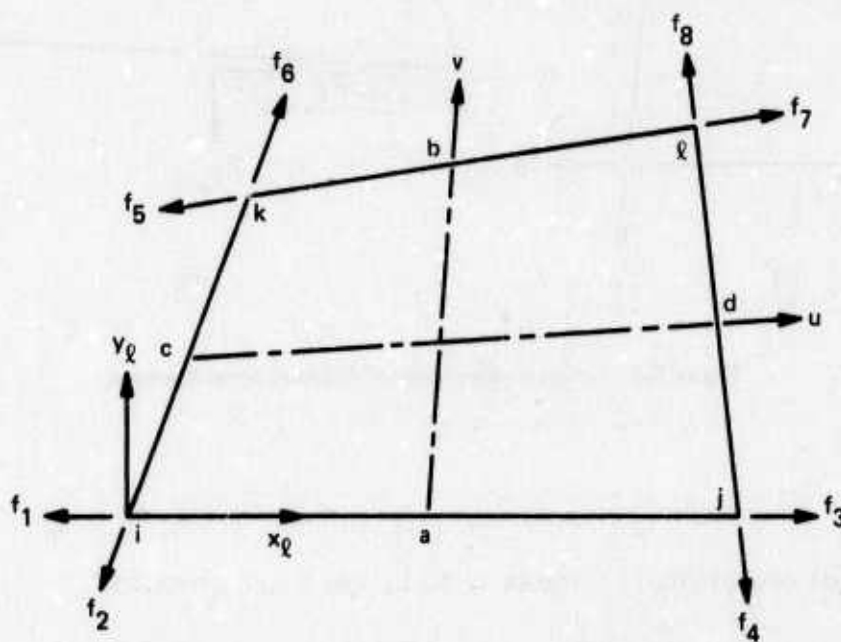


Figure E-2. Quadrilateral Element Geometry and Corner Forces

Figure E-3 shows the two free bodies formed by making the cuts described above. The forces shown on the cut sections are in the u, v axis system and are in equilibrium with the corner forces. They are first transformed to stress resultants in the same oblique axis system. As the two shear stress resultants thus obtained are not necessarily equal, a single shear stress resultant is determined by taking their average. Additional transformations then yield the stress resultants in the local-element-axis system or property-axis system, which ever is required. The details of this whole process are described below.

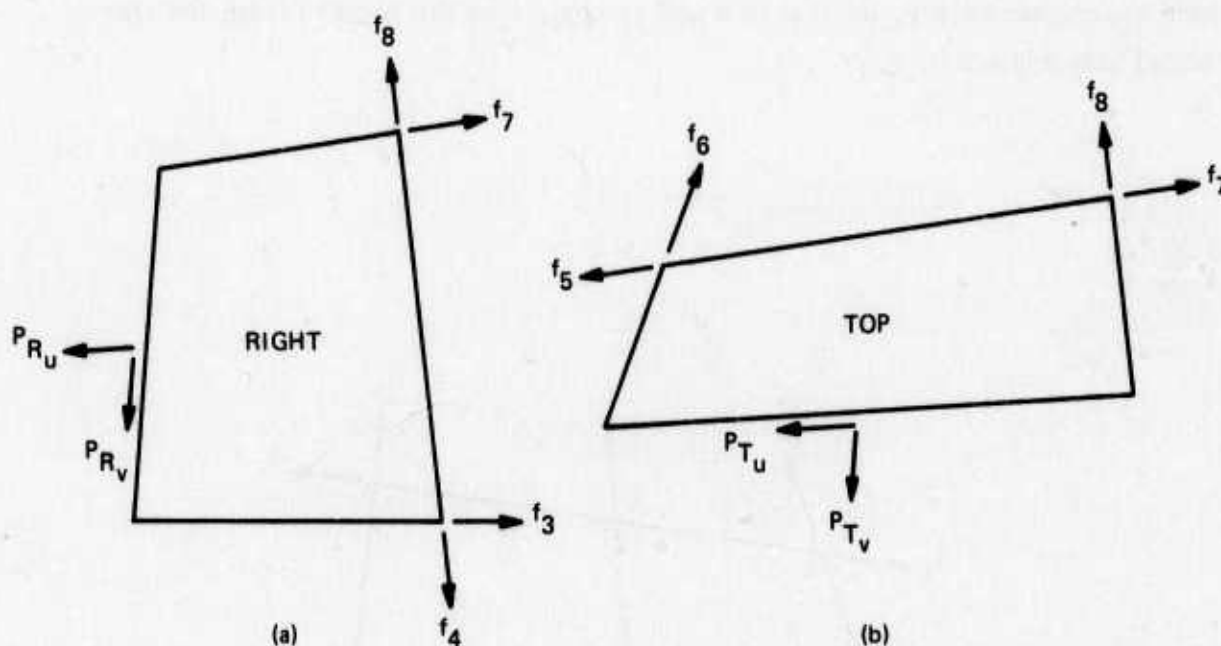


Figure E-3. Forces on Quadrilateral Element Cross Sections

The global coordinates of points a, b, c, and d are given by:

$$x_a = \frac{x_i + x_j}{2}, \quad y_a = \frac{y_i + y_j}{2}, \quad z_a = \frac{z_i + z_j}{2}$$

etc.

These coordinates are used in determining the direction cosines of unit vectors, \vec{u} and \vec{v} , along the u and v axes, in the manner shown in Appendix C. The direction cosines of the unit vector w along the w axis are determined from the relation:

$$\vec{w} = (\vec{u} \times \vec{v}) / L_w$$

where L_w is the length of the vector product in the numerator. These direction cosines form the elements of a matrix, $[\bar{D}]$, for the transformation of forces in the u, v, w system to the global axis system, represented as follows:

$$[\bar{D}] = \begin{bmatrix} d_{u_x} & d_{v_x} & d_{w_x} \\ d_{u_y} & d_{v_y} & d_{w_y} \\ d_{u_z} & d_{v_z} & d_{w_z} \end{bmatrix}$$

The resolution of the element corner forces and kick forces into the global axis system requires the generation of a table of direction cosines for forces defined in eight directions, as follows:

Direction No.	From Point	To Point
1	i	j
2	i	k
3	j	<i>l</i>
4	k	<i>l</i>
5	i	m
6	j	n
7	k	o
8	<i>l</i>	p

where m, n, o, and p are additional nodes introduced to define the directions of the kick forces, as discussed in Appendix A. The table of direction cosines for these eight directions is written as follows:

1	2	3	4	5	6	7	8
d_{1x}	d_{2x}	d_{3x}	d_{4x}	d_{5x}	d_{6x}	d_{7x}	d_{8x}
d_{1y}	d_{2y}	d_{3y}	d_{4y}	d_{5y}	d_{6y}	d_{7y}	d_{8y}
d_{1z}	d_{2z}	d_{3z}	d_{4z}	d_{5z}	d_{6z}	d_{7z}	d_{8z}

When nodes m, n, o, and p are not defined, the directions of the kick forces are taken to be the normals to the adjacent edges of the element, and may be obtained by forming the vector products of the vectors along these edges.

The corner forces and kick forces acting on the free bodies shown in Figure E-3 are now resolved into components in the directions of the global axes and are summed. The resulting forces acting on the free body shown in Figure E-4(a) are then given by:

$$\begin{Bmatrix} P_{R_x} \\ P_{R_y} \\ P_{R_z} \end{Bmatrix} = \begin{bmatrix} d_{1x} & -d_{3x} & d_{4x} & d_{3x} & d_{6x} & d_{8x} \\ d_{1y} & -d_{3y} & d_{4y} & d_{3y} & d_{6y} & d_{8y} \\ d_{1z} & -d_{3z} & d_{4z} & d_{3z} & d_{6z} & d_{8z} \end{bmatrix} \begin{Bmatrix} f_3 \\ f_4 \\ f_7 \\ f_8 \\ k_2 \\ k_4 \end{Bmatrix}$$

where the direction cosines are taken from the table above, the corner forces are as shown in Figure E-2, and the kick forces k_1 , k_2 , k_3 , and k_4 are applied at nodes i, j, k, and ℓ , as shown in Figure 30. The forces acting on the free body shown in Figure E-4(b) are similarly given by:

$$\begin{Bmatrix} P_{T_x} \\ P_{T_y} \\ P_{T_z} \end{Bmatrix} = \begin{bmatrix} -d_{4x} & d_{2x} & d_{4x} & d_{3x} & d_{7x} & d_{8x} \\ -d_{4y} & d_{2y} & d_{4y} & d_{3y} & d_{7y} & d_{8y} \\ -d_{4z} & d_{2z} & d_{4z} & d_{3z} & d_{7z} & d_{8z} \end{bmatrix} \begin{Bmatrix} f_5 \\ f_6 \\ f_7 \\ f_8 \\ k_3 \\ k_4 \end{Bmatrix}$$

The two sets of forces determined in this way are introduced into the following relations, to yield the reactions on the cross sections shown in Figure E-4:

$$\begin{Bmatrix} P_{R_u} \\ P_{R_v} \\ P_{R_w} \end{Bmatrix} = \begin{bmatrix} \overline{D} \end{bmatrix}^{-1} \begin{Bmatrix} P_{R_x} \\ P_{R_y} \\ P_{R_z} \end{Bmatrix}$$

$$\begin{Bmatrix} P_{T_u} \\ P_{T_v} \\ P_{T_w} \end{Bmatrix} = \begin{bmatrix} \overline{D} \end{bmatrix}^{-1} \begin{Bmatrix} P_{T_x} \\ P_{T_y} \\ P_{T_z} \end{Bmatrix}$$

and the stress resultants in the u and v coordinates are determined from them by the relations:

$$N_u = P_{R_u} / L_{ab}$$

$$N_v = P_{T_v} / L_{cd}$$

$$N_{uv} = \frac{1}{2} \left\{ (P_{R_v} / L_{ab}) + (P_{T_u} / L_{cd}) \right\}$$

A rectangular coordinate system of axes x', y' is now defined in the u, v plane such that x' is coincident with the u-axis, as shown in Figure E-4. It can be seen from Figure E-4 that the transformation of the stress resultants from the u, v system to the x', y' system is given by:

$$N_{x'} = N_u \operatorname{cosec} \theta + N_v \cos \theta \cot \theta + 2 N_{uv} \cot \theta$$

$$N_{y'} = N_v \sin \theta$$

$$N_{x'y'} = N_v \cos \theta + N_{uv}$$

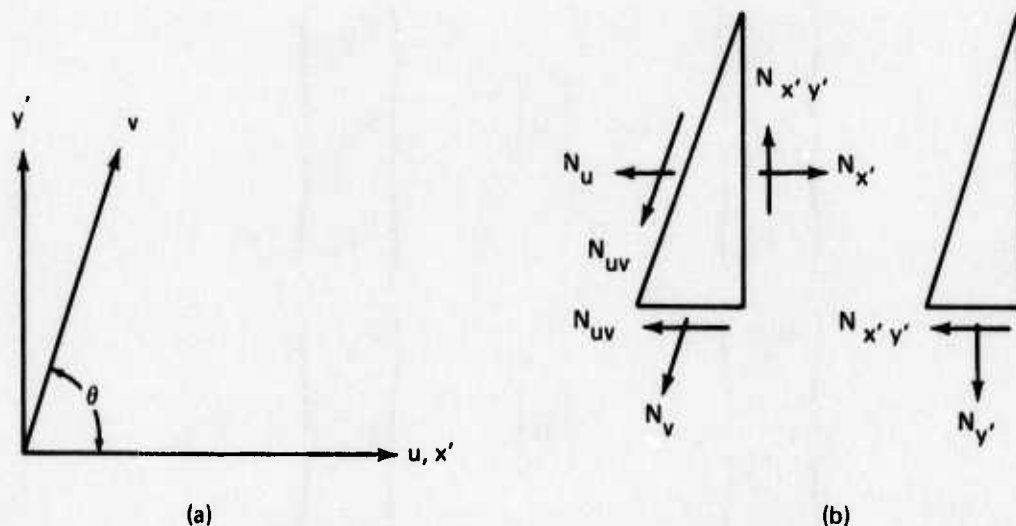


Figure E-4. Transformation from Oblique to Rectangular Coordinates

It is now necessary to transform these stress resultants from the x', y' system to the local-element axes x_l and y_l . For this purpose, we assume that the x_l and y_l axes lie in the u, v plane, and we determine the angle α between the x_l and x' axes, where α is positive for a positive rotation of the x_l and y_l axes into the x' and y' axes about \vec{w} in the right-hand sense. The determination of α is made in the same way as that of the angle β in Appendix C. That is, the projection of \vec{ij} into the x', y' plane is first obtained, and the angle between that projection and the x' axis is then determined. The transformation of stress resultants is then given by:

$$\begin{Bmatrix} N_{x_l} \\ N_{y_l} \\ N_{z_l y_l} \end{Bmatrix} = [B] \begin{Bmatrix} N_{x'} \\ N_{y'} \\ N_{x'y'} \end{Bmatrix}$$

where $[B]$ is defined in Appendix D-2, with α replacing β .

If the stress resultants in the property axis system are desired, an additional transformation is carried out, this time using the angle β in the transformation matrix $[B]$.

Appendix F

MATHEMATICAL RELATIONS USED IN STRESS CONSTRAINT RESIZING
OF COMPOSITES

F.1 DETERMINATION OF STRESS RATIO FOR COMPOSITE ELEMENT LAYERS

In the resizing of composite elements to satisfy stress constraints, it is first necessary to determine components of the stress resultant for the whole laminate in property axes, as described in Subsection 2.2.3 and Appendix E. The relationship between these stress resultant components and strain components must then be established. This relationship is written in the form:

$$\begin{Bmatrix} N_x^p \\ N_y^p \\ N_{xy}^p \end{Bmatrix} = [S] \begin{Bmatrix} \epsilon_x^p \\ \epsilon_y^p \\ \gamma_{xy}^p \end{Bmatrix} \quad (F-1)$$

where $[S]$ is the matrix of stiffness coefficients for the whole laminate. The stiffness of the matrix material is partially neglected in the determination of $[S]$, as discussed below. This has the effect of yielding strains and, consequently, stresses in the layers, as though the fibers were carrying practically the whole load. It should be noted that this procedure requires that the composite be fiber-controlled, as discussed in Subsection 2.2.3.

The matrix $[S]$ is formed by first forming the stiffness matrices for the individual layers in fiber axes, transforming them to property axes, and summing them, as follows:

$$[S] = \sum_{i=1}^n \ell_i t_i^T [D]_i [Q]_i [D]_i \quad (F-2)$$

where the summation is over all n layers in the laminate, subscript i refers to the i th layer, and:

- l is the number of laminae in the layer
- t is the thickness of an individual lamina
- $[D]$ is as defined in Equation (F-9), with ϕ_i replacing β , where ϕ_i is defined in Figure 40
- $[Q]$ is the matrix of stiffness coefficients in the stress-strain relation

The matrix $[Q]$ is as defined in Equation (F-13), but now with the stiffness of the matrix material effectively eliminated by setting $E_{22} = G_{12} = \nu_{12} = \nu_{21} = 0$. Thus:

$$[Q] = \begin{bmatrix} E_{11} & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad (F-3)$$

where E_{11} is the stiffness coefficient in the fiber direction for the layer under consideration. While E_{11} includes some effect of the matrix, this effect is normally small in composites in which the fiber material is much stiffer than the matrix material.

Solution of Equation (F-1) yields the strain components of the whole laminate, and, as all layers experience the same nodal displacement, these strain components also apply to the individual layers. These strain components are now transformed to fiber axes in each layer, as follows:

$$\begin{Bmatrix} \epsilon_x^f \\ \epsilon_y^f \\ \gamma_{xy}^f \end{Bmatrix} = [D] \begin{Bmatrix} \epsilon_x^p \\ \epsilon_y^p \\ \gamma_{xy}^p \end{Bmatrix} \quad (F-4)$$

where, again, $[D]$ is as defined in Equation (D-9), with ϕ_i replacing β .

Introduction of Equation (F-3) into the relation:

$$\begin{Bmatrix} f \\ \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = [Q] \begin{Bmatrix} f \\ \epsilon_x \\ \epsilon_y \\ \gamma_y \end{Bmatrix} \quad (\text{F-5})$$

yields the stress in the fiber direction:

$$\sigma_x^f = E_{11} \epsilon_x^f \quad (\text{F-6})$$

The stress in the fiber direction is determined in this way in each layer, and its absolute value is divided by the corresponding allowable stress (tension or compression as appropriate), to determine the stress ratio for layer resizing.

F.2 APPLICATION OF MICROBUCKLING FAILURE CRITERION

Using the components of the stress resultant for the whole laminate, the algebraically smaller of the two principal stress resultants is determined as follows:

$$N_{pc} = \frac{1}{2} \left(N_x^p + N_y^p - \sqrt{(N_x^p - N_y^p)^2 + 4 N_{xy}^p} \right) \quad (\text{F-7})$$

This is done for all loading conditions, and the largest negative value, $(N_{pc})_{\max}$, is introduced into the relation:

$$t = - \frac{(N_{pc})_{\max}}{G_{z_{av}}} \quad (\text{F-8})$$

where $G_{z_{av}}$ is a weighted average of the values of the microbuckling allowable G_z for the individual layers. $G_{z_{av}}$ is determined as follows:

$$G_{z_{av}} = \frac{\sum_{i=1}^n G_{z_i} \ell_i t_i}{\sum_{i=1}^n \ell_i t_i} \quad (F-9)$$

where the summation is over all layers, subscript i refers to the i th layer, and the weighting is seen to be done on the basis of layer thickness. The quantities ℓ_i and t_i are as defined in Subsection F.1, and the values of ℓ_i correspond to the layup existing at the beginning of each cycle in the cyclic process described in Subsection 2.2.3.

Equation (F-8) yields a laminate thickness required to satisfy the microbuckling failure criterion. If this thickness exceeds the laminate thickness existing at that point, following the application of all other criteria, the additional thickness required is made up by adding laminae to layer number 1, always rounding up to determine the required number of laminae.

This procedure requires that the user decide in advance which layer he would like to build up in order to satisfy the microbuckling failure criterion, and to designate that layer as layer number 1. Usually that layer will be the one with fibers in or near the principal direction of loading, such as the spanwise direction in the case of a wing or tail surface.

Appendix G

DERIVATION OF MATHEMATICAL RELATIONS AND CRITERIA USED IN THE DEFLECTION-CONSTRAINT ALGORITHM

G.1 UNIFORM DERIVATIVES AS AN OPTIMALITY CRITERION IN DEFLECTION-CONSTRAINT RESIZING

Let a structure with n design variables w_i ($i = 1, 2, \dots, n$), where w_i is the weight of the i th element, be subjected to a single-deflection constraint in a single loading condition, represented by:

$$\delta = \delta_{\text{desired}} \quad (\text{G-1})$$

and to no other constraint of any type. The total weight of the structure is given by:

$$W = \sum_{i=1}^n w_i \quad (\text{G-2})$$

Equation (G-1) will normally define a curved surface in the n -dimensional design space, w_1, w_2, \dots, w_n , and equation (G-2) will define planes in that space for constant values of W . If a minimum-weight design exists that satisfies the deflection constraint and involves nonzero values of all the design variables, there will be at least one constant-weight plane that is tangent to the deflection-constraint surface, and the point of tangency will define that design.

The components of the gradient to the constant-weight plane are given by:

$$\frac{\partial W}{\partial w_i} = 1 \quad (i = 1, 2, \dots, n) \quad (\text{G-3})$$

and the components of the gradient to the deflection-constraint surface are given by

$\frac{\partial \delta}{\partial w_i}$ ($i = 1, 2, \dots, n$). At the point of tangency of the two surfaces, the two gradient vectors will necessarily be colinear, and, as $\frac{\partial W}{\partial w_i}$ ($i = 1, 2, \dots, n$) are all equal, it follows that $\frac{\partial \delta}{\partial w_i}$ ($i = 1, 2, \dots, n$) will also be equal.

It should be noted that a minimum-weight design, satisfying the deflection constraint and involving nonzero values of all the design variables, does not always exist. As an example, consider the simple structure shown in Figure G-1. Members AB and CD are elastic rods, while BD is a rigid beam. A constraint is to be applied on the angular displacement Θ of BD. The only design variables are w_1 and w_2 , the weights of members AB and CD, respectively. It is easily shown that the angular displacement Θ depends only on the relative values of w_1 and w_2 , in such a way that a given value of Θ can be maintained, while w_1 and w_2 both become vanishingly small. This is illustrated in Figure G-2, where the constraint curve for constant Θ is shown in the two-dimensional design space. Clearly, the minimum weight design, in the absence of other constraints, is at point O, the origin of the design space. If minimum gage constraints are applied to w_1 and w_2 , as shown by the dashed lines in Figure G-2, the minimum-weight design will be at point A.

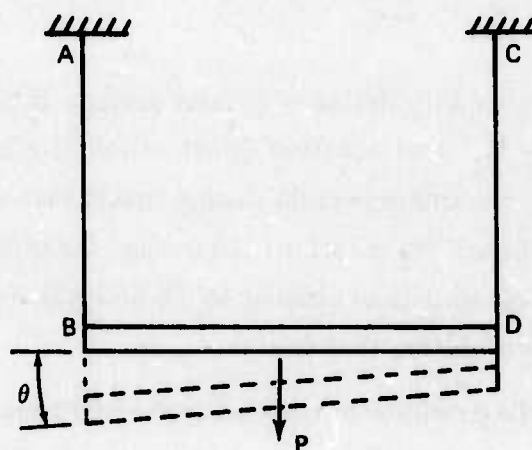


Figure G-1. Simple Structure Subjected to Angular Displacement Constraint

In the general case, there may be design variables that have a zero value in the minimum-weight design. If these variables are regarded as inactive variables, the uniform-derivative criterion for optimality will still be valid, if it is applied only to the remaining, or active, variables. In the application of the deflection-constraint algorithm described in Subsection 2.3.1, the inactive variables will be the ones that yield a negative value for the quantity under the radical in Equation (2.6),

and they can be permitted to reduce to zero. If minimum-gage constraints are applied, the inactive variables can be permitted to reduce to their minimum-gage values.

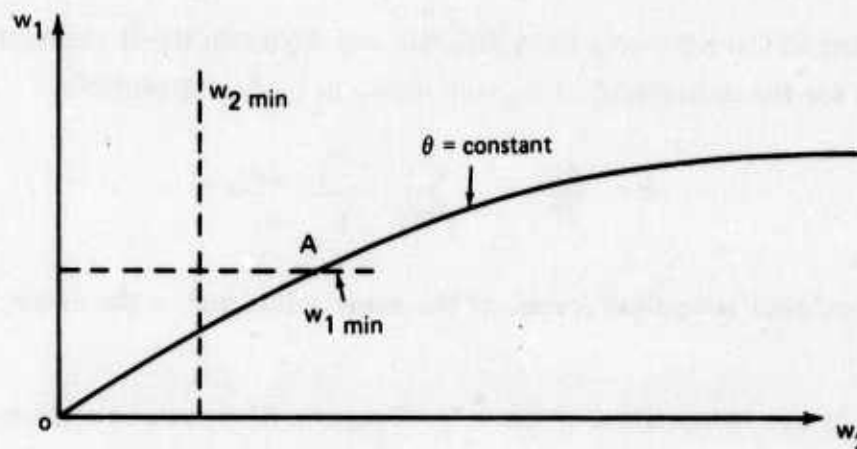


Figure G-2. Design Space for Structure of Figure G-1

When stress constraints are present, in addition to the deflection constraint, the uniform-derivative criterion is not rigorously applicable. It can, however, be regarded as an approximate criterion when applied to the design variables that are governed by the deflection constraint, and should yield a design that is near minimum in weight. This situation is similar to that for fully-stressed design, where it is recognized that the design obtained is minimum in weight only in an approximate sense.

G.2 THE RECURRENCE RELATION IN DEFLECTION-CONSTRAINT RESIZING

In a statically determinate structure, the internal loads are independent of the design variables. In a truss structure, for example, these loads are the loads p_i ($i = 1, 2, \dots, n$) in the members for a given applied loading. The strain or elongation in each member will be inversely proportional to its cross-sectional area, and, as

the strain energy of each member is proportional to the product of the internal load and elongation, the strain energy of the whole structure can be written in the form:

$$U = \sum_{i=1}^n \frac{a_i}{w_i} p_i \quad (G-4)$$

where w_i is the weight of the i th member and a_i is a constant for the i th member.

Application of Castigliano's first theorem and Equation (G-4) yields the following expression for the deflection, at a given joint, in a given direction:

$$\delta = \frac{\partial u}{\partial P} = \sum_{i=1}^n \frac{a_i}{w_i} \frac{\partial p_i}{\partial P} \quad (G-5)$$

where P is an externally applied force, at the same joint, and in the same direction as δ .

As the p_i 's are independent of the w_i 's, Equation (G-5) can be written in the form:

$$\delta = \sum_{i=1}^n \frac{b_i}{w_i} \quad (G-6)$$

where the b_i 's are quantities that are independent of the w_i 's.

If a constraint is applied to δ , Equation (G-6) will describe a constraint surface for the constraint value of δ . The components of the gradient to this surface are given by:

$$\frac{\partial \delta}{\partial w_i} = - \frac{b_i}{w_i^2} \quad (G-7)$$

At the point on this surface representing the minimum-weight design satisfying the constraint, the derivatives given by Equation (G-7) will be equal for all design variables, as explained in Subsection G.1. That is,

$$- \frac{b_i}{w_i^2} = K \quad (G-8)$$

where \tilde{w}_i ($i = 1, 2, \dots, n$) are the member weights in the minimum-weight design and K is a constant.

Equation (G-8) may be substituted into Equation (G-7) to yield:

$$\frac{\partial \delta}{\partial w_i} = K \frac{\tilde{w}_i^2}{w_i^2} \quad (G-9)$$

or:

$$\tilde{w}_i = \sqrt{\frac{\frac{\partial \delta}{\partial w_i}}{K}} w_i \quad (G-10)$$

When any design satisfying the constraint is given, Equation (G-10) may be used to determine the minimum-weight design satisfying the constraint. It is necessary, in the process, to determine a value of K that, when substituted into Equation (G-10), will yield a design satisfying the constraint.

Equation (G-10) is seen to have the same form as Equation (2.6) in Subsection 2.3.1.

Although Equation (G-10) was derived for a statically-determinate truss, it can be shown to be equally applicable to any statically determinate structure. It is not directly applicable to statically indeterminate structures, where the internal loads are dependent upon the design variables, in that it will not directly yield a minimum-weight design. However, because the internal loads are usually not highly sensitive to variations in the design variables, Equation (G-10) has been found to be useful as a recurrence relation, when applied in an iterative procedure, as described in Subsection 2.3.1. That procedure has been found to converge to uniform values of the derivatives for the active variables.

G.3 DETERMINATION OF DEFLECTION DERIVATIVES

The relations needed in the determination of the partial derivatives of the generalized deflection, δ , subject to constraint, with respect to the element weights, w_i , are now derived. Starting with the basic equation relating the applied loads $\{P\}$ to the associated nodal displacements $\{\delta^P\}$:

$$\{P\} = [K] \{\delta^P\} \quad (G-11)$$

the partial derivatives of the applied loads with respect to the weight, w_i , of the i th element, are formed:

$$\frac{\partial \{P\}}{\partial w_i} = \left\{ \frac{\partial [K]}{\partial w_i} \right\} \{\delta^P\} + [K] \frac{\partial \{\delta^P\}}{\partial w_i} \quad (G-12)$$

Both sides of Equation (G-12) are identically zero, because the applied loads are not functions of the design variables, so that:

$$[K] \frac{\partial \{\delta^P\}}{\partial w_i} = - \frac{\partial [K]}{\partial w_i} \{\delta^P\} \quad (G-13)$$

The generalized deflection is now written in the form:

$$\delta = \{\bar{Q}\}^T \{\delta^P\} \quad (G-14)$$

where $\{\bar{Q}\}$ is a vector in which weighting coefficients are placed in the locations corresponding to the degrees of freedom of the deflection they multiply.

The partial derivative of δ with respect to w_i is given by:

$$\frac{\partial \delta}{\partial w_i} = \{\bar{Q}\}^T \frac{\partial \{\delta^P\}}{\partial w_i} \quad (G-15)$$

If $\{\bar{Q}\}$ is regarded as a "virtual load" vector, a corresponding "virtual displacement" vector, $\{\delta^q\}$, may be determined by solution of the equation:

$$[K] \{\delta^q\} = \{\bar{Q}\} \quad (G-16)$$

Premultiplication of both sides of Equation (G-13) by $\{\delta^q\}^T$ yields:

$$\{\delta^q\}^T [K] \frac{\partial \{\delta^P\}}{\partial w_i} = - \{\delta^q\}^T \frac{\partial [K]}{\partial w_i} \{\delta^P\} \quad (G-17)$$

From Equations (G-16) and (G-15), it can be seen that the left-hand side of Equation (G-17) is simply $\frac{\partial \delta}{\partial w_i}$. Furthermore, if the elements of the stiffness matrix,

$[K]$, are linear functions of the design variables, w_i , as they usually are, and $[k_i]$ is the stiffness matrix of the i th element for unit value of w_i , $\frac{\partial [K]}{\partial w_i}$ is seen to be equal to $[k_i]$. Thus, Equation (G-17) may be written in the form:

$$\frac{\partial \delta}{\partial w_i} = - \{\delta^q\}^T [k_i] \{\delta^p\} \quad (G-18)$$

where $\{\delta^q\}$ and $\{\delta^p\}$ are compressed to contain only the degrees of freedom associated with member i .

It is seen that the determination of the derivatives of the generalized deflection, with respect to the element weights involves the following steps:

- (a) Determine the nodal deflections, $\{\delta^p\}$, due to the applied loading condition in which the deflection constraint is applied, by solution of Equation (G-11).
- (b) Form the virtual load vector, $\{\bar{Q}\}$, for the generalized deflection subject to constraint, and solve Equation (G-16) for the virtual displacements, $\{\delta^q\}$.
- (c) Substitute $\{\delta^q\}$ and $\{\delta^p\}$, suitably compressed, into Equation (G-18), to determine the required partial derivative for each element.

G.4 TREATMENT OF A VIOLATED INEQUALITY CONSTRAINT AS AN EQUALITY CONSTRAINT.

As discussed in Subsection 2.3.2, the deflection-constraint mode is entered, in the case of an inequality deflection constraint, only if that constraint is violated by the design existing at the end of the stress-constraint mode. The deflection constraint is then treated as an equality constraint.

The question arises as to whether the design subsequently achieved after convergence in the deflection-constraint mode is necessarily as low in weight as any other design on the feasible side of the constraint boundary, but not on it, and satisfying the same optimality criteria.

In addressing this question, we consider first the design existing at the end of the stress-constraint mode and assume that all members are either fully-stressed or at a minimum gage. This design is represented by point A in Figure G-3, which is a plot of the subject deflection versus total structure weight. It is assumed to be a

minimum-weight design under the constraints imposed, at least in an approximate sense, accepting that this is not rigorously true for a fully-stressed design.

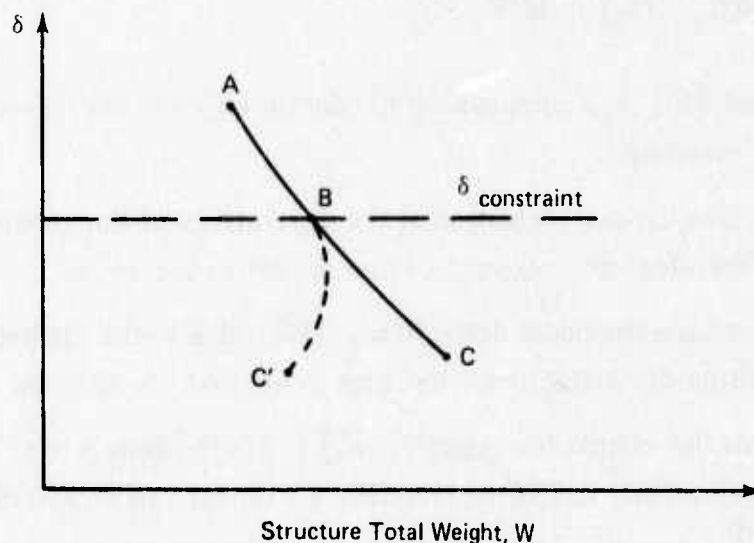


Figure G-3. Satisfaction of an Inequality Deflection Constraint

Point B represents the design that satisfies the deflection constraint as an equality constraint (it is on the constraint boundary) and also satisfies the optimality criteria. That is, all members fall into two groups. In one group, all members are either fully-stressed or are at minimum gage, while all members in the other group are governed by the deflection constraint and have uniform values of the derivative, $\partial\delta/\partial w_i$.

On the ground that the imposition of an additional constraint cannot result in a decrease in structure weight, and should result in an increase in weight if that constraint was violated prior to its imposition, it is concluded that the design at B is heavier than the design at A.

The question that is being posed is whether there are other designs, such as that at point C', which are on the feasible side of the constraint boundary and are lower in weight than the design at B, or whether the trajectory of design points satisfying the

optimality criteria must be of the form BC. If we consider the group of members that are governed by the deflection constraint, and consider small changes in their weight, Δw_i ($i = 1, 2, \dots, m$), the corresponding change in the subject deflection is given by:

$$\Delta \delta = \sum_{i=1}^m \frac{\partial \delta}{\partial w_i} \Delta w_i \quad (G-19)$$

Because of uniformity of the derivatives, Equation (G-19) can be written:

$$\begin{aligned} \Delta \delta &= \frac{\partial \delta}{\partial w_i} \sum_{i=1}^m \Delta w_i \\ &= \frac{\partial \delta}{\partial w_i} \Delta W \end{aligned} \quad (G-20)$$

where ΔW is the corresponding change in total structure weight, due only to weight changes in the group of members governed by the deflection constraint.

It should now be recalled that, in moving from point A to point B, the sign of the derivatives of those members selected for deflection-constraint resizing should be such that a change in δ , in the desired direction, is associated with an increase in w_i . Accordingly, it is seen from Equation (G-20) that a change $\Delta \delta$ toward the feasible side of the constraint boundary must be accompanied by a net increase in weight of the members governed by the deflection constraint. This will be true not only at point B, but also in moving from any point to any other nearby point along the trajectory of design points satisfying the optimality criteria, that is, from B toward C (or C').

The question posed now reduces to the question of whether the increase in gage and weight of the members governed by the deflection constraint can permit a reduction in gage of the fully-stressed members and a corresponding net decrease in weight of those members that exceeds the net increase in the weight of the former group, to produce a decrease in the total structure weight. In the case of statically determinate structures, the stress in any member is independent of the gage of any other member, so that the fully-stressed members cannot decrease in weight. In the case of statically indeterminate structures with weak coupling between members (in terms of the effect

of change in gage of any one member on the stress in any other member), it can be expected that there will generally not be a decrease in total structure weight in the situation described above.

While a proof does not appear to be available for the general case, establishing that movement away from the constraint boundary into the feasible design space cannot result in a decrease in total structure weight - and perhaps such a proof cannot even be made - it can be expected that it is a reasonable assumption to make in most practical cases.

Appendix H

LIST OF PROGRAM SUBROUTINES

A list of the subroutines presently in ASOP-3 is presented below. The symbols following each subroutine name indicate whether the subroutine is unchanged (U), slightly changed (S), or extensively changed (E) from ASOP-2, or whether it is a new subroutine (N).

AONE (U)	DLIMIT (E)	INPUT (S)
ARACE (U)	DPRINT (U)	INVERS (S)
ASOP (S)	DRATIO (E)	IPROPA (N)
ASTACK (S)	DSCALE (E)	LAMRES (N)
AVGSTN (N)	EL1 (U)	LDGEN (S)
AWFIN (S)	EL2 (U)	LENTH (U)
AWRITE (S)	EL3 (U)	LNGTH (N)
BETAİN (N)	EL5 (U)	LOADIN (U)
BMTRAN (S)	EL6 (U)	MATRAL (N)
BOUND (U)	EL8 (S)	MAX (U)
CARDIN (S)	EL15 (not being used)	MEMBIN (E)
CARDS (S) (not present in CDC version)	EL16 (not being used)	MEMDEF (N)
	ENMMPY (U)	MEMWT (N)
CHGCMP (N)	EXDEF (U)	MILTOT (U)
CHGMEM (N)	EXMOUT (N)	MULT (U)
CHKANG (N)	FCAPG (E)	NEWLAM (N)
CNEWT (N) (not being used)	FILTAP (U)	NEWT (N)
COMPLN (N)	FINDAT (N)	NUREAD (U)
COSINE (U)	FLOAT (U)	PACK (U)
CRINT (S)	GEOBC (S)	PAGES (S)
CTSUM (N)	GETDIM (U)	PATNIK (S)
DEFCON (E)	GETQ (N)	PRINT2 (U)
DINIT (U)	GETROW (U)	PROCES (U)
DINTK (U)	HOTDOT (U)	PUNCHO (U)
	INDEX (U)	PUNLIB (U)

PUTLAB	(U)	RTAPE	(E)	UNITWT	(S)
PUTROW	(U)	SBMAIN	(S)	UNPACK	(U)
QBAC	(U)	SECOND	(U)	VECTOR	(U)
QBISOL	(U)	SETUP	(S)	WARPS	(S)
QBSOL	(U)	SKIPIN	(U)	WEIGHT	(E)
QCHOL	(U)	SPJT	(U)	WORK	(N)
QFACT	(U)	SPLITS	(S)	WRTCMP	(N)
QFIN	(U)	SREVN2	(U)	WRTMOS	(N)
QFOR	(U)	SSTRES	(U)	WRTSTR	(N)
QSIN	(U)	STABIL	(U)	WTSUM	(N)
QFSOL	(U)	STNEU	(N)	ZBAR	(N)
QIBAC	(U)	STRFC	(N)	ZBEAM	(N)
QPASS	(U)	STRRAT	(N)	ZIP	(U)
REPABL	(U)	SUPPRT	(U)	ZQUAD	(N)
REREAD	(U)	TABDG	(U)	ZQUADC	(N)
RESIZE	(E)	TARDIV	(N)	ZTRI	(N)
REVERS	(S)	THERM	(not being used)		

The following ASOP-2 subroutines have been deleted:

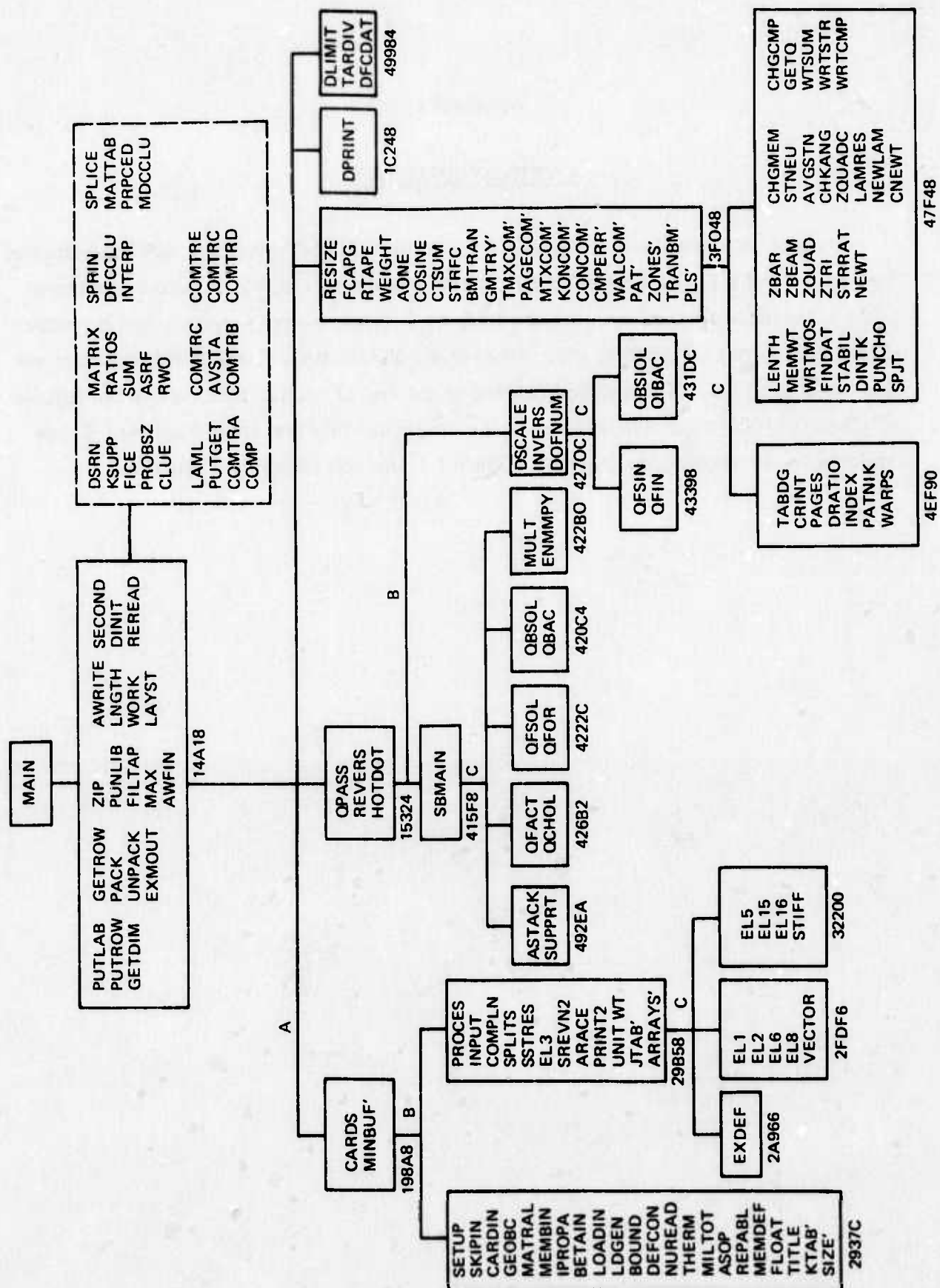
COMPEL
ESSE
LAMOPT
LMPROP
MINMAX

NUSIZE
ORTHOG
STRANS
TRISIZ

Appendix I

OVERLAY CHARTS

Charts, showing the overlay structure of the ASOP-3 program, are presented in Figures I-1 and I-2, for IBM and CDC systems respectively. In Figure I-1, names listed in the block framed by a broken line, as well as primed names listed elsewhere on the chart, are COMMON blocks rather than subroutines. COMMON blocks are not listed in Figure I-2. The number entered at the end of each link in both of the figures indicates the total core requirement of the program when that link is active. These numbers are in hexadecimal form in Figure I-1 and octal form in Figure I-2.



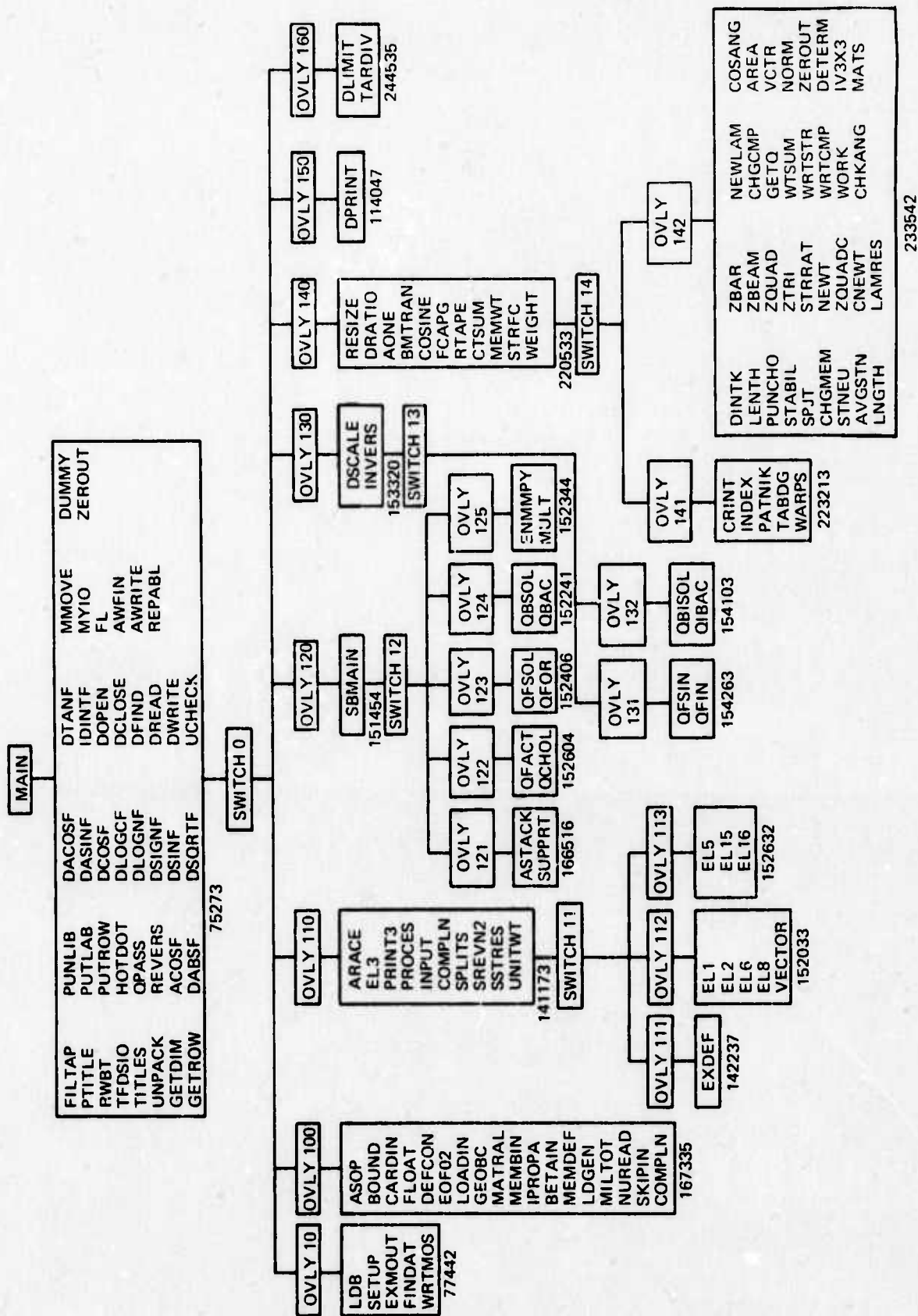


Figure I-2. Overlay Chart for ASOP-3 in CDC Systems

REFERENCES

1. Razani, R. , "The Behavior of the Fully-Stressed Design of Structures and its Relationship to Minimum-Weight Design", AIAA Journal, Vol. 3, No. 12, December 1965, pp. 2262-2268.
2. Dwyer, W. J. , Emerton, R. K. , and Ojalvo, I. U. , "An Automated Procedure for the Optimization of Practical Aerospace Structures. Vol I - Theoretical Development and User's Information. Vol. II - Programmer's Manual". AFFDL-TR-70-118, April 1971.
3. Dwyer, W. J. , "An Improved Automated Structural Optimization Program", AFFDL-TR-74-96, September 1974.
4. Dwyer, W. , Rosenbaum, J. , Shulman, M. , and Pardo, H. , "Fully Stressed Design of Airframe Redundant Structures", presented at the Air Force Second Conference on Matrix Methods in Structural Mechanics, October, 1968.
5. Zienkiewicz, O. C. , The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill, London, 1967.
6. Przemieniecki, J. S. , Theory of Matrix Structural Analysis, McGraw-Hill, New York, 1968.
7. Garvey, S. J. , "The Quadrilateral Shear Panel", Aircraft Engineering, May 1951.
8. Tsai, S. W. , "Strength Characteristics of Composite Materials", NASA CR-224, April 1965, pp. 5-8.
9. Sandhu, R. S. , "A Survey of Failure Theories of Isotropic and Anisotropic Materials", AFFDL-TR-72-71, September 1972, pp. 19 and 22.
10. Jones, R. M. , Mechanics of Composite Materials, Scripta Book Company, Washington, D. C. , 1975.
11. Suarez, J. A. , Hadcock, R. N. , and Whiteside, J. B. , "The Influence of Local Failure Modes on the Compressive Strength of Boron-Epoxy Composites", Composite Materials; Testing and Design (Second Conference), American Society of Testing and Materials, ASTM STP 497, 1972, pp. 237-256.

12. Lager, J. R., and June, R. R., "Compressive Strength of Boron-Epoxy Composites", J. Composite Materials, January 1969, pp. 48-56.
13. Berke, L., "Convergence Behavior of Iterative Resizing Procedures Based on Optimality Criteria", Air Force Flight Dynamics Laboratory Technical Memorandum 72-1-FBR, September 1972.
14. Wilkinson, K., Lerner, E., and Taylor, R. F., "Practical Design of Minimum-Weight Aircraft Structures for Strength and Flutter Requirements", J. of Aircraft, Vol. 13, No. 8, August 1976, pp. 614-624.
15. Berke, L., and Khot, N. S., "Use of Optimality Criteria Methods for Large-Scale Systems", presented at AGARD Lecture Series on Structural Optimization, 10-18 October 1974, AFFDL-TM-74-70-FBR, April 1974.
16. Khot, N. S., Venkayya, V. B., and Berke, L., "Optimum Design of Composite Structures with Stress and Deflection Constraints", AIAA Paper No. 75-141, presented at AIAA 13th Aerospace Sciences Meeting, Pasadena, California, January 20-22, 1975.
17. Taig, I. C., and Kerr, R. I., "Optimization of Aircraft Structures with Multiple Stiffness Requirements", AGARD Second Symposium on Structural Optimization, Milan, Italy, April 2-4, 1973, AGARD-CP-123, pp 16-1 - 16-14.
18. Wilkinson, J. H., "The Algebraic Eigenvalue Problem", Clarendon Press, Oxford, 1965.
19. Ashton, J. E., Halpin, J. C., and Petit, P. H., Primer on Composite Materials: Analysis, Technomic Publishing Co., Stamford, Conn., 1969.
20. MacNeal, R. H., (Editor), "The NASTRAN Theoretical Manual" (Level 15), NASA SP-22(01), April 1972, p. 5.8-28.
21. Balderes, T., DyReyes, V., and Iaccarino, S., "A Description of the Average Stress Program (S50) - Theoretical and Users' Manual", Grumman Aerospace Corporation, Report No. SAR-75-1, June 18, 1975.

THIS REPORT HAS BEEN DELIMITED
AND CLEARED FOR PUBLIC RELEASE
UNDER DOD DIRECTIVE 5200.20 AND
NO RESTRICTIONS ARE IMPOSED UPON
ITS USE AND DISCLOSURE.

DISTRIBUTION STATEMENT A

APPROVED FOR PUBLIC RELEASE;
DISTRIBUTION UNLIMITED.